# Tecplot®

## User's Manual

### Version 9.2, Release 1

Amtec Engineering, Inc.
Bellevue, Washington

September, 2002

# Contents

# CHAPTER 1 What's New in Tecplot Version 9.0

Tecplot Version 9.0 renders images using OpenGL, which dramatically enhances Tecplot's capabilities, especially in the area of three-dimensional data visualization. This chapter highlights the broad improvements to Tecplot and details how using Tecplot Version 9.0 differs from using past versions of Tecplot.

## 1.1. New Capabilities

The most significant improvements to Tecplot Version 9.0 concern 3-D plotting, image import and export, and curve-fit augmentation. An overview of each is provided in the following sections.

### 1.1.1. 3-D Upgrades

Tecplot's new 3-D capabilities include:

- **Fast rendering:** OpenGL allows Tecplot to utilize today's fast graphics cards.
- **Unified view controls:** Magnification, rotation, and translation modes have been integrated into mouse controls, eliminating the time spent hopping from tool to tool. Time-saving mouse and keyboard shortcuts allow greater efficiency for creating plots.
- **Translucency:** Pack more information into your plots by making your iso-surfaces, slices, streamtraces, and other zonal surfaces translucent. This is true translucency, not the pseudo-translucency of previous versions that was based on a form of dithering. An example is shown in Figure 1-1.

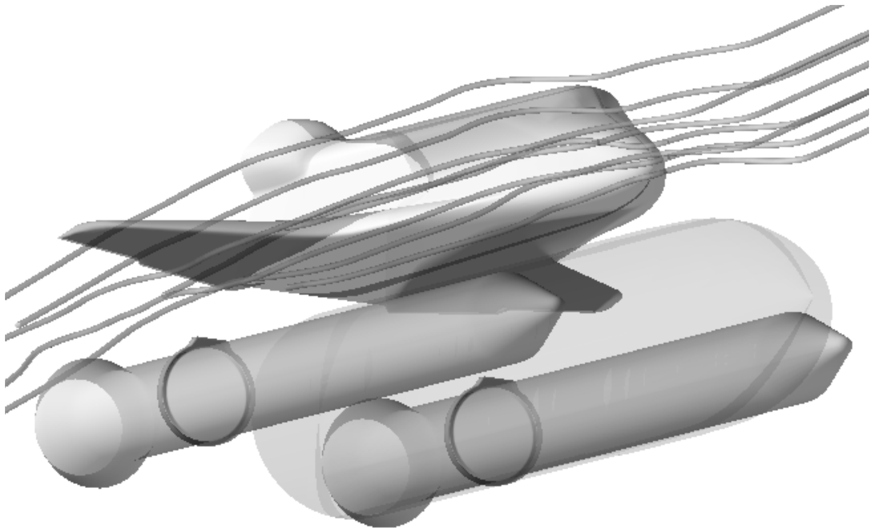


**Figure 1-1.** An example of a translucent plot created with Tecplot Version 9.0.

- **Slicing:** A suite of new tools allow you to interactively place and display slices for 3-D volume data sets without having to first extract them to zones. Flooded contours, shading, mesh lines, vectors and scatter symbols instantly appear on your slices. Simultaneously slice multiple planes, or sweep a slice through a volume to explore your data. The slices may be created for constant X-, Y-, Z-, I-, J-, or K-planes. An example is shown in Figure 1-2.
- **Iso-surfaces:** Increase and decrease iso-surface values in 3-D data sets to discover information quickly. You may show one or more values at the same time, and like Tecplot's slicing capabilities, you do not have to extract to zones to display them with lighting effects.
- **Streamtraces:** Just point-and-click to place streamtraces in 3-D volumes on slices or iso-surfaces within volumes. Your streamtraces are rapidly rendered, and just as rapidly removed if you desire. Information on your slices, iso-surfaces, and streamtraces is saved to your layout file so you can recreate them in seconds.
- **True color:** Contour flooding and light source shading on 3-D surfaces are now rendered in vivid colors to create stunning plots and animations.

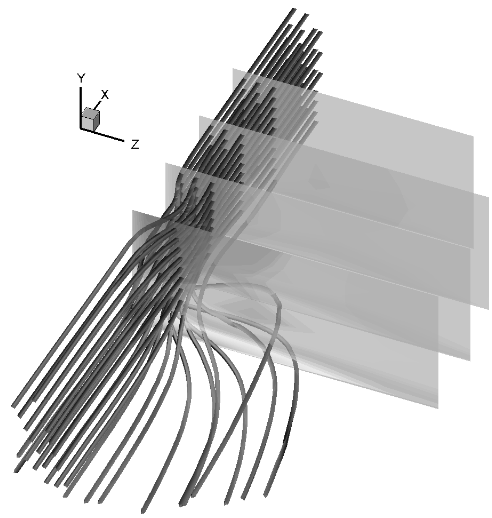



**Figure 1-2.** An example of interactive information discovery utilizing Tecplot's slicing abilities.

### 1.1.2. Import and Export

Tecplot's new import and export abilities include:

- New CGNS and Fluent data loaders.
- A new Image Loader so you may load your logo as a set of geometries.
- Export vibrant 24-bit raster images in true color, or reduce to 256 colors for compactness.
- Exported images may be any resolution independent of your monitor.

### 1.1.3. Curve-Fits

Tecplot, through the Add-on Developer's Kit, gives you the ability to create your own curve-fits. Customized curve-fits using your own proprietary algorithms enhance Tecplot. Also, with an Amtec-supplied curve-fit add-on, you can interactively define curves with up to eight degrees of freedom.

## 1.2. Changes from Version 8.0

Tecplot Version 9.0 uses OpenGL for 3-D on-screen imaging and for exporting raster images. With Version 9.0, all details are redrawn with each movement. To fully realize the 3-D performance of Version 9.0 we strongly recommend that you install a powerful OpenGL-accelerated graphics card on your computer.

To maintain responsiveness when viewing extremely large data sets, you may switch to a trace image (an approximate wire-frame view) during operations such as rotation and translation. Specify the trace option by setting the Draw Level for 3D View Changes to Trace on the Performance Options dialog, which is accessed from the sidebar.

**Note:** For optimum performance on Windows, go to your Display properties by clicking with your right mouse button on your desktop, then choose Properties. Make sure that "Show window contents while dragging" is turned off under "Plus!" (Windows NT) or "Effects" (Windows 98 or 2000).

### 1.2.1. 3-D Images

The following changes have been made as to how Tecplot Version 9.0 handles objects in 3-D:

- In Version 8.0 there was only one global light source color. Now each light source shaded object may be a different color.
- The 3-D lighting effects on light source shaded objects is now specified in the new Effects page of the Plot Attributes dialog. These effects may be applied to flooded contour surfaces, shaded surfaces, iso-surfaces, slices, and streamtraces. (Previously, to have a lighting effect on a contour-flooded object, you had to turn on both contours and shade, and set the Contour layer to be pseudo-translucent.)
- IJK Attributes have been changed to Volume Attributes. For each zone, you may choose which surfaces to plot (surfaces, planes, and so forth), which points to plot, and which 3-D objects to plot (whether to draw streamtraces, slices and iso-surfaces for that zone).
- There is no need to extract streamtraces, slices or iso-surfaces to view them with correct sorting or with more advanced surface plot styles. They are displayed immediately with mesh, shading, and/or flooded contours.
- Iso-surfaces may be viewed independently of a zone's contour lines. They are controlled by their own dialog. You may choose to see iso-surfaces for all contour levels, or for up to three specified values of the contour variable.

- A new 3D Slice Details dialog allows you to display a specified number of slices on X-, Y-, or Z-planes. Ordered data also has the option of displaying slices for constant I, J, or K. A new tool has been added to the sidebar allowing you to begin creating and displaying slices, or change the position of existing slices. When it is selected, you can change the plane of the slices by typing the appropriate letter, such as Z for a Z-plane. The starting plane may be moved by clicking the desired location. Shift-click on your data to add or move the ending slice plane, and press any number on the keyboard to set the number of intermediate slices. Slices can display flooded contours, vectors, meshes, shading, and boundaries. When extracting slices, choose to extract only from the surface zones or only from the volume zones.
- Screen images and exported bit-based images can have true translucency in 3-D. It can be specified as any value from 1-99 percent. Translucency may be applied to streamtraces, isosurfaces, slices, shaded surfaces, and contour flooded surfaces.
- Finite-element volume data performance is greatly improved. The new default is to draw only the outer surfaces, so you do not have to extract a finite-element boundary to view it. Hidden surface removal for 3-D finite-element data has also been improved.
- Rotation has been smoothed and is faster and more accurate.

### 1.2.2. Printing and Exporting Images

The following changes have been made as to how Tecplot Version 9.0 prints and exports images:

- Exported raster images may have true translucency.
- Plots exported in a vector-based format such as PostScript and Windows Metafile may appear different than the screen image. Translucency is only supported in raster image output. Hidden surfaces may show some minor artifacts at intersecting surfaces.
- When printing, new render options allow you to print a bit-based image and specify its size. A print preview option is available, to view how a plot would appear as an exported vector-based file.
- Appending to existing files with Tecplot's animation export formats (Raster Metafile and Audio-Visual Interleaved) is no longer supported. Appending is only supported while creating the file; you cannot stop the animation and restart it in the middle at some future time. If you wish to make animations and later concatenate them together you may create animations as separate Raster Metafile movies and later concatenate them together. The rmtoavi utility allows you to convert the Raster Metafiles to AVI files.

### 1.2.3. Performance

OpenGL, along with new algorithms, has lead to increased performance in the following areas:

- Adding or removing streamtraces or iso-surfaces does not cause a recalculation of the other objects, so it is much faster.
- Many internal calculations such as slicing, blanking interpolation, and streamtrace integration are faster.
- Creating streamtraces for 2- and 3-D plots is now faster and more accurate.
- More colors are available for contour flooding. The number of colors available is now tied to the maximum contour levels allowed. There is also a continuous color flooding option which smoothly varies the color for 3-D plots.
- The change from one color to the next on multi-color and color-flooded streamtraces, streamrods, and streamribbons, is now accurate with the change of the contour variable, instead of an abrupt "block" change perpendicular to the sides of the streamtrace.
- The new Slice tools allow you to extract the slices you have created in your plot.
- You may display the boundary for a finite-element quadrilateral or finite-element triangle zone.
- Tecplot allows use of non-traditional ordered data such as J-, K-, IK-, and JK-ordered.
- You may now probe exclusively on non-zone objects (like streamtraces, slices, and iso-surfaces) using the Alt key. For example, if you have a slice in a volume plot, you may use the Alt with the Streamtrace tool to place a streamtrace directly on the slice. You could also use Alt with the Selector tool to select the slice, then use the Quick Edit dialog to add or remove mesh, contour, and so forth on the slice.

### 1.2.4. XY Curve-Fits

The following changes have been made to curve-fitting:

- Curve-fit types are now located on the Curves page of the Plot Attributes dialog, instead of the Lines page, as in Version 8.0.
- In addition to the standard curve-fits, you may also choose the Extended option, which lists all curve-fits added to Tecplot as add-ons.

### 1.2.5. Keyboard and Mouse Operations

The following operations are allowed while using most sidebar tools with a mouse:

- For a three-button mouse:

  **Middle button:** Zoom in/out.

  **Right button:** Translate.
- For a two-button mouse:

  **Ctrl-Right button:** Zoom in/out.

  **Right button:** Translate.

Further enhancements include:

- You may now drag the mouse when using the Contour Add tool. In addition, if you hold the Ctrl key down you may adjust the location of an existing contour line or iso-surface.
- When adding streamtraces you can press R, D, V, or S on your keyboard to switch to ribbons, rods, volume lines, or surface lines. Pressing a number on your keyboard changes the number of streams to place in a rake.
- All interactive rotations using the mouse will be smooth. The step size now only applies to clicking the rotate options on the sidebar.
- When using the Zoom tool you can click once with left mouse button to center the zoom around the point you click.
- Alternate, viewer-centric rotate and zoom operations are now available. An Alt-middle mouse button, or Alt-Ctrl-right mouse button, while dragging with your mouse will increase or decrease the view distance instead of view width. When in one of the Rotate modes, adding Alt while dragging your mouse will cause the viewer, rather than the object, to rotate. These new modes are useful for flyby-like examinations of your 3-D plot.

### 1.2.6. Interface Navigation Changes

Functionality changes due to menu and dialog modifications are detailed below.

- **3D Light Source Color Dialog:** This dialog has been removed from the Workspace menu. It is no longer an option.
- **Contour Colormap Adjustments:** Colormap Override options are on the new Advanced Options dialog, which is accessed from the new Contour Coloring Options dialog.
- **Lift Fractions for Vector, Geometry and Scatter:** These have moved from the Field menu’s 3D Details dialog to the Field menu’s new Advanced 3D Control dialog. They are now called Lift Fractions for Line, Symbol, and Tangent.
- **Lighting Effects:** To add a lighting effect to a contour flooded zone, go to the Field menu’s Contour Attributes dialog and set Use Lighting to Yes. Then go to the Effects page and set Lighting Effect to Paneled or Gouraud. This may also be done for streamtrace rods or rib-

bons on the Field menu's Streamtrace Details dialog, for slices on the Field menu's 3D Slice Details dialog, and for iso-surfaces on the Field menu's 3D Iso-Surface Details dialog.

- **Light Source Position:** This option has been moved from the Field menu's 3D Details dialog to the Field menu's new 3D Light Source dialog.
- **Orthographic to Perspective 3-D Images:** This ability has moved from the Field menu's 3D Details dialog to the View menu's new 3D View Details dialog. (The parameters have also been changed.)
- **Streamtraces:** Because there are several new options for displaying streamrods and streamribbons, the Field menu's Streamtrace Details dialog has a new page for Rod/Ribbon. (Windows users should use the tab page arrows to access the Integration page.)
- **Z-Clipping:** This ability has moved from the Field menu's 3D Details dialog to the Style menu's new 3D Depth-Blanking dialog. Cells are blanked along an imaginary plane parallel to the surface of the monitor screen.

### 1.2.7. Macro Language Changes

To accommodate the changes in Tecplot Version 9.0 there are many new and updated macro commands. In most cases, Version 8.0 macros will run without modification. The biggest exceptions are macros which create Raster Metafile or AVI animation files. These must be modified to work with Tecplot Version 9.0.

### 1.2.8. Tecplot Version 8.0 Layout Suggestions

If a 3-D layout created in Tecplot Version 8.0 looks very dark, try moving the light source direction. Do this using the 3D Light Source dialog accessed from the Field menu. You may also try increasing the amount of background light. This option, which is also available on the 3D Light Source dialog, along with Intensity and Surface Color Contrast, gives you excellent control over the coloring of your plots.

On all 3-D zones with both contour flooding and shading, turn off the Shade zone layer on the sidebar, then change the Lighting Effect to Paneled or Gouraud using the Field menu's Effect Attributes dialog.

# CHAPTER 2 Getting Started

Tecplot is a powerful tool for visualizing a wide range of technical data. It offers XY-plotting, 2- and 3-D surface plots in a variety of formats, and 3-D volumetric visualization, combined within an easy to learn point-and-click interface. This chapter describes Tecplot's interface and goes through the basic procedures for creating a variety of graphics. We will use data sets included with Tecplot for the examples in this chapter, and many examples in the rest of the book, so you may create these plots.

## 2.1. Starting Tecplot

The following sections describe how to start Tecplot on Windows or UNIX systems.

### 2.1.1. Windows

On Windows operating systems you start Tecplot from the Start button, or from an icon on your desktop.

To start Tecplot from the Start button:

1. Click Start, then select Programs.
2. Select the Tecplot 9.0 folder.
3. Click on Tecplot.

Following the opening banner, the Tecplot window appears, as shown in Figure 2-1.

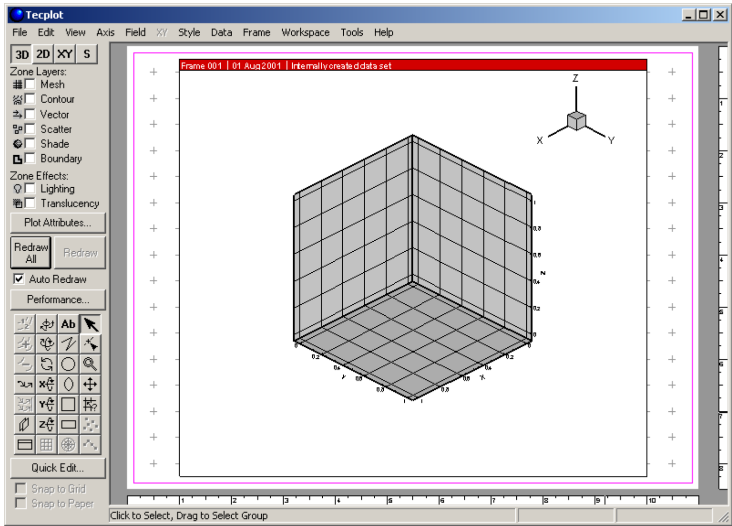


**Figure 2-1.** The Tecplot window under Windows.

### 2.1.2. UNIX

On UNIX systems, Tecplot is typically installed by a system administrator, who then makes it available to end users. You then run Tecplot by typing:

```
tecplot
```

at the shell prompt. The opening banner appears, followed by the Tecplot window, as shown in Figure 2-1.

The directory in which Tecplot is installed, on any platform, is called the Tecplot home directory. You should know the absolute path of this directory and set your **TEC90HOME** environment variable to point to it. The Tecplot home directory includes numerous example files referred to throughout this manual; by working with these files you can quickly gain proficiency with Tecplot's features. A list of the example data files and their features appears in Appendix D, "List of Example Files."

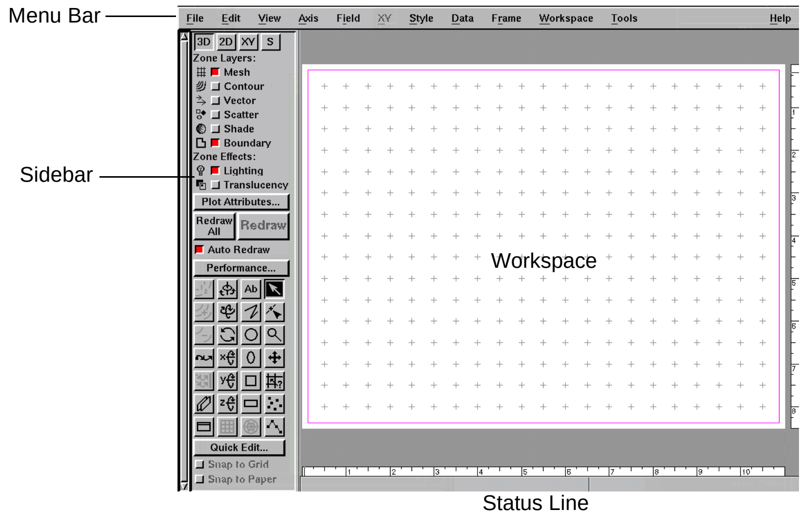



**Figure 2-2.** The Tecplot window in Motif, showing the four main regions: menu bar, sidebar, workspace and status line.

# 2.2. The Interface

Figure 2-2 shows the Tecplot window as it appears at startup with no initial data set. There are four main regions in the Tecplot window: the menu bar, the sidebar, the workspace, and the status line.

## 2.2.1. The Menu Bar

The menu bar, shown in Figure 2-3, offers rapid access to most of Tecplot's features, which are controlled primarily through dialogs, secondary windows that contain one or more controls for managing various aspects of the plot.

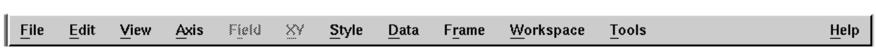



**Figure 2-3.** The menu bar.

Tecplot's features are organized into the following menus:

- **File:** Use the File menu to control reading and writing of data files and plot layouts, printing and exporting of plots, recording and playing back macros, setting and saving your configuration preferences, and exiting Tecplot.
- **Edit:** Use the Edit menu to control cutting, copying, pasting, and clearing objects, as well as pushing and popping them (which can change the order in which Tecplot draws them). The Edit menu also contains an option for adjusting data points.

  Tecplot's Cut, Copy, and Paste options work only within Tecplot. If you are operating the Windows version of Tecplot and want to place a graphics image of your layout into other word processing or graphics software, you can do so using the Copy Plot to Clipboard option.
- **View:** Use the View menu to control the point of view of your data, including the scale, viewed range, and 3-D rotation. You can also use the View menu to copy and paste views between frames.
- **Axis:** Use the Axis menu to control the axes in XY, 2D, and 3D frame modes.
- **Field:** Use the Field menu to control field plots in 2D and 3D frame modes (mesh, contour, vector, scatter, shade, streamtrace, 3-D iso-surface, 3-D slice, and boundary plots).
- **XY:** Use the XY menu to control XY-plotting.
- **Style:** Use the Style menu to control text, geometries (polylines, circles, squares, ellipses, and rectangles), data labeling and blanking features. The Style menu also has options for copying and pasting stylesheet files.
- **Data:** Use the Data menu to create, manipulate, and examine data. Types of data manipulation available in Tecplot include simple zone creation, interpolation, triangulation, and creation or alteration of variables by means of FORTRAN-like equations.
- **Frame:** Use the Frame menu to create, edit, and control frames.
- **Workspace:** Use the Workspace menu to control the attributes of your workspace, including the color map, paper grid, display options, and rulers.
- **Tools:** Use the Tools menu to run any Quick Macros you may have defined, or to create simple animations of your plots. (Add-ons other than data loaders and extended curve-fits are also accessed through the Tools menu.)
- **Help:** Use the Help menu to get quick help on features. By selecting About Tecplot. you can obtain specific information about your license. The Help menu also gives you access to information about the add-ons you have loaded.

### 2.2.2. The Sidebar

Tecplot's sidebar accesses the most frequently used controls for plotting. Many take the form tools, which control the behavior of the pointer in the workspace. Additional controls determine frame mode, which layers are active, and snap modes. The controls are organized in the following functional clusters, as shown in Figure 2-4:

- **Frame Modes.**
- **Zone/Map Layers.**
- **Zone Effects.**
- **Plot Attributes.**
- **Redraw All-Redraw-Auto Redraw.**
- **Performance.**
- **Tools.**
- **Quick Edit/Object Details.**
- **Snap Modes.**

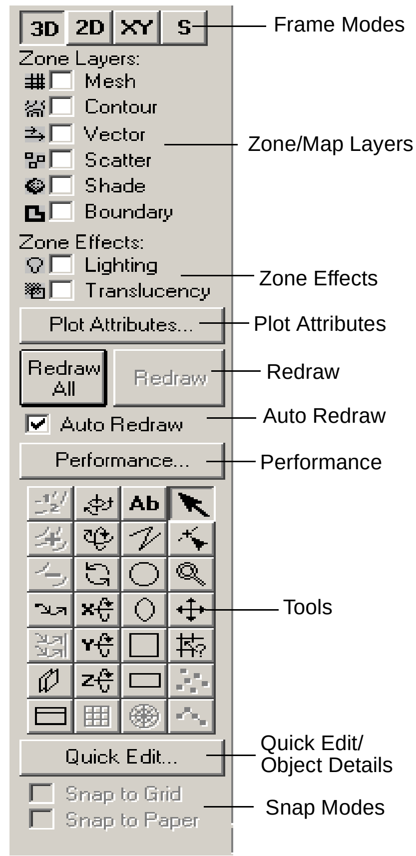



**Figure 2-4.** The Tecplot sidebar.

**2.2.2.1. Frame Modes.** A frame mode determines, in a broad sense, what type of plot can be drawn in the current frame. There are four:

- **3D:** Create 3-D plots of surfaces and volumes.
- **2D:** Create 2-D field plots, which will often be plots of some variable by location on a plane.
- **XY:** Create XY-plots, such as plots of independent versus dependent variables.
- **S (Sketch):** Create plots without data such as drawings, flow charts, and viewgraphs.

The frame mode, combined with a frame's data set, the active plot layers and their associated attributes, defines the plot. Each frame mode represents just one view of the data.

**2.2.2.2. Zone Layers/Map Layers.** A zone layer is one way of representing a frame's data set. The complete plot is the sum of all the active layers, axes, text, geometries, and other elements added to the basic data plotted in the layers. There are six zone layers for 2D and 3D frame mode, four map layers for XY frame mode, and no zone layers in Sketch frame mode.

The six zone layers for 2D and 3D frame modes, as shown in Figure 2-4, are:

- **Mesh:** The Mesh zone layer plots the lines connecting the data points within each zone.
- **Contour:** The Contour zone layer plots contours, which in Tecplot can be either lines having a constant value, or the region between these lines, or both.
- **Vector:** The Vector zone layer plots the direction and magnitude of vector quantities.
- **Scatter:** The Scatter zone layer plots symbols at the location of each data point.
- **Shade:** The Shade zone layer may be used to shade each zone with a specified solid color, or to add light-source shading to a 3-D surface plot. Used in conjunction with the Lighting zone effect you may set Paneled or Gouraud shading. Used in conjunction with the Translucency zone effect you may create a translucent surface for your plot.
- **Boundary:** The Boundary zone layer plots the zone boundaries for ordered data.

The four map layers in XY mode, shown in Figure 2-5, are:

- **Lines:** This map layer plots a pair of variables, X and Y, as a set of line segments or a fitted curve.
- **Symbols:** This map layer plots a pair of variables, X and Y, as individual data points represented by a symbol you specify.
- **Bars:** This map layer plots a pair of variables, X and Y, as a horizontal or vertical bar chart.
- **Error Bars:** This map layer plots error bars in any of several formats.

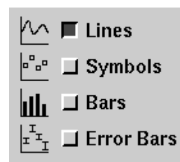



**Figure 2-5.** XY map layers.

**2.2.2.3. Zone Effects.** In 3D frame mode the check boxes shown in Figure 2-6 appear: Lighting; Translucency. Only shaded and flooded contour surface plot types are affected by Lighting and Translucency.

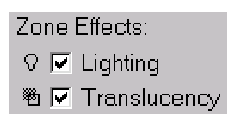



**Figure 2-6.** Zone Effects options.

**2.2.2.4. The Plot Attributes Button.** The Plot Attributes button calls up the Plot Attributes dialog, which allows you to modify the appearance of each zone.

**2.2.2.5. The Redraw Buttons.** Tecplot does not automatically redraw the plot after every change, unless you select the Auto Redraw check box. The Redraw buttons, as shown in Figure 2-7, allow you to keep your plot up to date.


**Figure 2-7.** Redraw buttons.

- **Redraw:** Redraws only the current frame.
- **Redraw All:** Redraws all frames. Shift-Redraw All causes Tecplot to completely regenerate the workspace.

**2.2.2.6. Auto Redraw.** The Auto Redraw check box allows you to continuously update your plot.

**2.2.2.7. The Performance Button.** The Performance button calls up the Display Performance dialog, where you may configure Tecplot's status line and performance options. The Display Performance dialog allows you to adjust Tecplot's performance to suit your individual needs. For further details, see Section 31.3, "Using the Display Performance Dialog."

**2.2.2.8. The Tool Buttons.** Each of the tools represented by a button is a mouse mode, which specifies the behavior of the mouse pointer anywhere in the workspace. There are 28 modes, which fall into the following 12 categories, as shown in Figure 2-8:

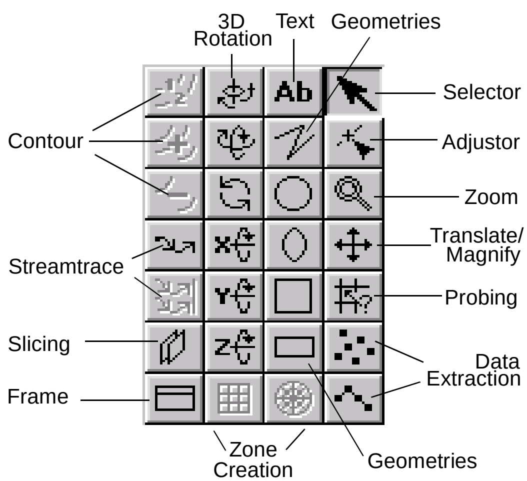



**Figure 2-8.** Sidebar tools and mouse modes.

- **Contour mouse modes.**
- **Streamtrace mouse modes.**
- **Slicing mouse mode.**
- **Frame mouse mode.**
- **Zone creation mouse modes.**
- **3-D rotation mouse modes.**
- **Text mouse mode.**
- **Geometry mouse modes.**
- **Mouse pointer modes:** Selector and Adjustor.
- **View mouse modes:** Zoom and Translate/Magnify.
- **Probe mouse mode.**
- **Data extraction mouse modes.**

**2.2.2.9. Enhanced Tool Operations.** Several of the sidebar tools offer mouse and keyboard shortcuts which can greatly speed Tecplot use, especially when working with large amounts of data. These are:

- **Contour tools:**

  **+:** Switch to Contour Add tool if you are using Contour Remove.

  **- :** Switch to Contour Remove tool if you are using Contour Add.

- **Contour Add tool:**

  **Click:** Place a contour line.

  **Ctrl-click:** Replace the nearest contour line with a new line.

  **Drag:** Move the new contour line.

- **Streamtrace Placement tool (3D frame mode only):**

  **D:** Switch to streamrods.

  **R:** Switch to streamribbons.

  **S:** Switch to surface lines.

  **V:** Switch to volume lines.

  **Alt-click/Alt-drag:** Determine the XYZ-location by ignoring zones and looking only at derived volume objects (streamtraces, iso-surfaces, slices).

  **1-9:** Change the number of streamtraces to be added when placing a rake of streamtraces.

- **Slicing tool:**

  **Click:** Place a start slice.

  **Drag:** Move the start slice.

  **Alt-click/Alt-drag:** Determine the XYZ-location by ignoring zones and looking only at derived volume objects (streamtraces, iso-surfaces, slices).

  **Shift-click:** Place the end slice

  **Shift-drag:** Move the end slice.

  **+:** Turns on the start slice if no slices are active, or turns on the end slice if slices are already active.

  **- :** Turns off the end slice if the end slice is active, or conversely, turns off the start slice if the end slice is not active.

  **I, J, K (ordered zones only):** Switch to slicing constant I-, J-, or K-planes respectively.

  **X, Y, Z:** Switch to slicing constant X-, Y, or Z-planes respectively.

  **0-9:** Numbers one through nine activate intermediate slices and set the number of intermediate slices to the number entered; zero turns off intermediate slices.

- **Zoom tool:**

  **Click:** Center a 200 percent magnification around the location of your click.

**2.2.2.10. Enhanced Mouse Operations.** The middle and right mouse buttons allow you to smoothly zoom and translate your data. Your middle mouse button (or Ctrl-right click) zooms smoothly, and your right mouse button translates data. This advanced functionality is available in:

- **All Contour Modes.**
- **Streamtrace Placement.**
- **Slicing.**
- **All 3-D Rotation Modes.**
- **All Geometry Modes (Except Polyline).**
- **Zooming.**
- **Translate/Magnify.**
- **Probing.**
- **Zone Creation.**

**2.2.2.11. The Details Button.** Immediately under the sidebar tools is a single button with a context-sensitive label, referred to as the Details button. Use this button to call up the dialog most directly applicable to your current action. When the currently selected tool is either the Selector (  ) or the Adjustor (  ), but no objects are selected in the workspace, the Details button is labeled Quick Edit. When either of those tools is selected and one or more objects are selected in the workspace, the label changes to Object Details. If any other tool is selected, the label changes to read Tool Details.

**2.2.2.12. The Quick Edit Button.** The Quick Edit button calls up the Quick Edit dialog, which you can use to make rapid changes to selected objects in the workspace.

**2.2.2.13. Snap Modes.** Snap modes, as shown in Figure 2-9, allow you to place objects precisely by locking them to the nearest reference point, either on the axis grid or on the workspace paper. There are two:

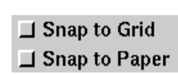



**Figure 2-9.** Snap mode buttons.

- **Snap to Grid.**
- **Snap to Paper.**

### 2.2.3. The Status Line

The status line, running along the bottom of the Tecplot window, gives “hover help.” When you move the mouse pointer over one of the sidebar tools, any button on the Quick Edit dialog, or over a menu item, it displays a brief description of the control. When you choose a tool, the help changes to a brief instruction for that tool. The status line also provides a variety of other information for specific purposes.

The configuration of the status line can be changed by selecting Interface from the Preferences sub-menu of the File menu.

### 2.2.4. The Workspace

The workspace, shown below, is the portion of your screen in which you create sketches and plots. All sketching and plotting is done inside a frame, which can be manipulated much like a process window. The current state of the workspace, including the sizing and positioning of frames, the location of the data files used by each frame, and all current plot attributes for all frames, makes up a Tecplot layout. By default, the workspace displays a representation of the paper Tecplot is set up to draw on, as well as a reference grid (for precise placement of frames on the paper) and rulers (for measuring frame and object sizes). The active frame, in which you are currently working, is on top. All modifications are made to the current frame.

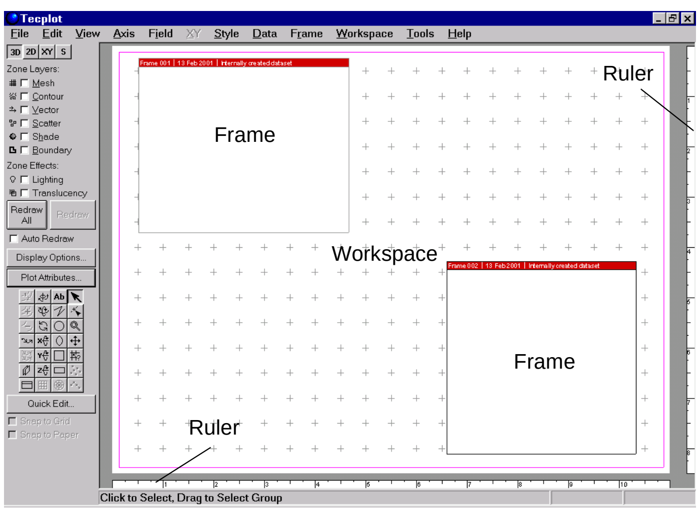



**Figure 2-10.** The Tecplot workspace.

## 2.2.5. Dialogs

Most dialogs use some combination of the following controls:

- **Buttons:** When you click on a button, Tecplot performs some action. Most dialogs have at least two buttons, Close and Help. Click Close to close the dialog. Click Help to display the Help system. Other common buttons are the Increase and Decrease buttons, and the sidebar buttons, previously described. Figure 2-11 shows the Increase and Decrease buttons as they appear in Motif and Windows systems.

**Figure 2-11.** Increase and Decrease buttons in Motif (left) and Windows (right).

- **Option Buttons:** Option buttons allow you to select only one of a set of mutually exclusive options; when you click an option, that option is selected and any previously selected option is deselected. Figure 2-12 shows option buttons as they appear in Motif and Windows systems.

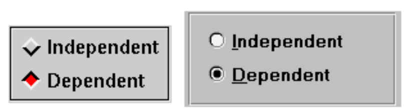



**Figure 2-12.** Option buttons in Motif (left) and Windows (right).

- **Check boxes:** Check boxes are Yes/No or On/Off switches. Typically, a check box names some option. If the check box is selected, the option is in effect. If the check box is not selected, the option is not in effect. Figure 2-13 shows check boxes as they appear in Motif and Windows systems.

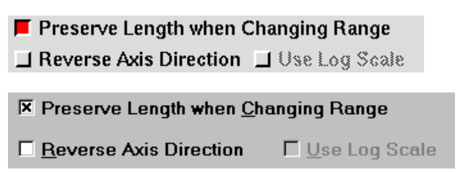



**Figure 2-13.** Check boxes in Motif (top) and Windows (bottom).

- **Text fields:** Use text fields to type information, such as file names, arbitrary parameters, or data values. Figure 2-14 shows text fields as they appear in Motif and Windows systems.

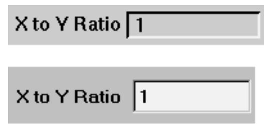



**Figure 2-14.** Text fields in Motif (top) and Windows (bottom).

- **Sliders:** Sliders, also known as scales, are controls that allow you to specify any value in a specific range by using the pointer to drag a thumb slider button back and forth or up and down along a scale. For fine control, you can use the arrow keys on your keyboard to move the thumb in small increments. Figure 2-15 shows sliders as they appear in Motif and Windows systems.

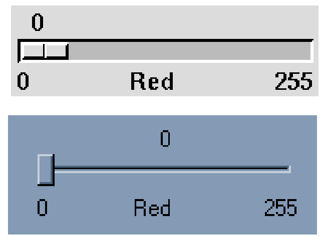



**Figure 2-15.** Sliders in Motif (top) and Windows (bottom).

- **Scrolled lists:** Scrolled lists, also known as list boxes, are lists of options, such as file names. Sometimes scrolled lists allow multiple selections, sometimes they are restricted to one selection only. Figure 2-16 shows scrolled lists as they appear in Motif and Windows systems.

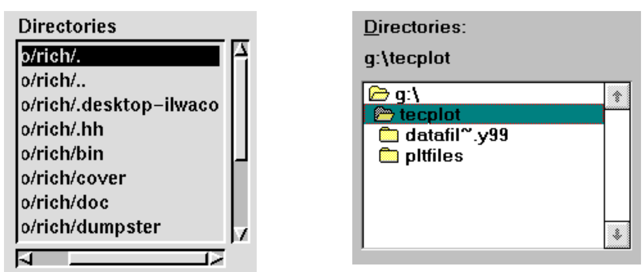



**Figure 2-16.** Scrolled lists in Motif (left) and Windows (right).

- **Drop-downs:** Any menu or list of options which becomes visible when you click at a particular spot is a drop-down. Thus, all the menus available from the menu bar and their submenus, as well as various types of option menus and drop-down lists, are called drop-downs. Figure 2-17 shows drop-downs as they appear in Motif and Windows systems.

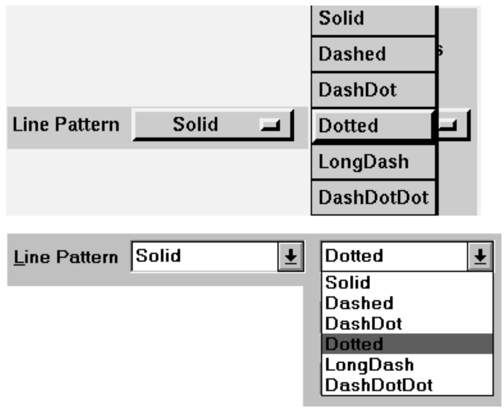



**Figure 2-17.** Drop-downs in Motif (top) and Windows (bottom).

### 2.2.6. File Dialogs

Each type of Tecplot file has at least two dialogs associated with it: one for opening files and one for saving files. All of these dialogs are very similar to each other, but they differ greatly depending on whether you are using Tecplot under Motif or under Windows. This section gives basic procedures for using these file dialogs under both Motif and Windows.

**2.2.6.1. Working with File Dialogs in Motif.** Figure 2-18 shows a typical Motif file dialog: the Open Layout dialog. Near the bottom of the dialog is a text field labeled Selection. If you know the complete path of the file you want to open or save, you can simply type it into this field and press Enter or click OK. At the top of the dialog is a text field labeled Filter (Name Search). You can use this field to specify a file name filter. Using a file name filter causes Tecplot to display all sub-directories of the current directory in the Directories scrolled list, as well as all files in the current directory ending with the extension **`.lay`** in the Files scrolled list. Dialogs for other file types have different default filters—for example, the data file dialogs have a filter that displays files with the extension **`.plt`**. The filter determines the initial path displayed in the Selection text field. To change the default file extensions, see Section 30.1.4., “Specifying Default File Name Extensions.”

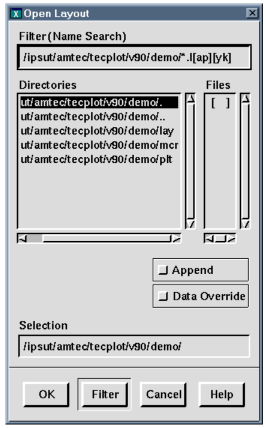



**Figure 2-18.** The Open Layout dialog in Motif.

You can supply a new filter by selecting the Filter text field and typing in new text. Typically, the only part of the filter you will change is the file path, so that you can list files in a different directory. Press Enter or click Filter to update your Files and Directories scrolled lists as well as the Selection text field.

You can also modify the filter by choosing a different directory from the Directories scrolled list. The directory specified by the current filter is displayed, along with its subdirectories and its parent directory (the line ending with "**..**"). Click on any directory in the list to make it the current filter directory, then press Enter or click Filter to update the Files and Directories scrolled lists as well as the Selection text field. Double-clicking on a directory name has the same effect.

When your filter shows the directory containing the file you want to open or save, you can select it in any of the following ways:

- Click on the file name in the Files scrolled list, then press Enter or click OK.
- Double-click on the file name in the Files scrolled list.
- Type the name of the file in the Selection text field, then press Enter or click OK. (The insertion point is initially set at the end of the filter path, so all you have to do is type in the file name.)

**2.2.6.2. Working with File Dialogs in Windows.** Figure 2-19 shows a typical Windows file dialog—the Open Layout dialog. In the lower half of the dialog is a text field labeled File

name. If you know the complete path of the file you want to open or save, you can type it into this field, then press Enter or click Open. You can also use this field to specify a file name filter. By default, the Open Layout dialog has a file name filter of `*.lay` and `*.lpk`, and the list of file names displays all files in the current directory ending with the extension `.lay` and `.lpk`. Dialogs for other file types have different default filters—for example, the data file dialogs have a filter that displays files with the extension `.plt` and `.dat`. To change the default file extensions, see Section 30.1.4., "Specifying Default File Name Extensions."

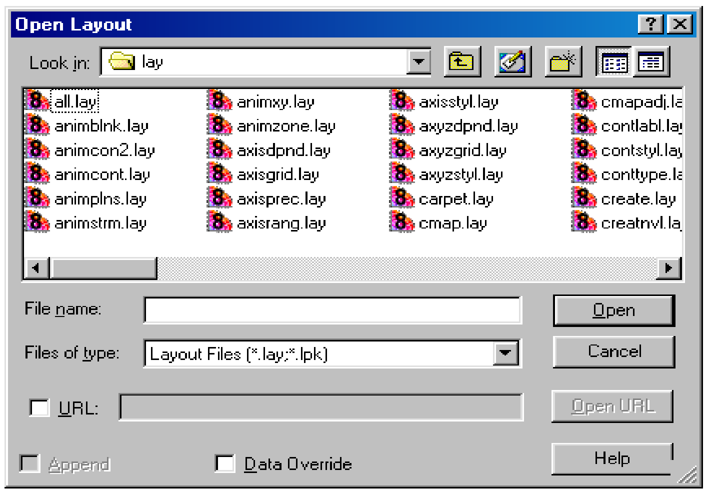



**Figure 2-19.** The Open Layout dialog in Windows.

You can supply a new filter by selecting the text in the File name text field and typing in new text. Press Enter to update the File name text field, the Look in drop-down, and its list of files and folders.

You can also modify the filter by choosing a different directory or folder from the Look in drop-down, or you can choose a folder in the field below the Look in drop-down, which displays files and folders in the drive or folder selected in the Look in drop-down. To move to the parent folder or directory of the current folder or directory, click Up One Level, at the right of the Look in drop-down. Clicking View Desktop will change the filter to your desktop. Clicking again will take you back to the folder or directory displayed before View Desktop was selected. Click on any directory in the list to make it the current filter directory and press Enter, or double-click the directory name to update the File name text field, the Look in drop-down and its list of files and folders.

When your filter shows the directory containing the file you want to open or save, you can select it in any of the following ways:

- Click on the file name in the scrolled list under the Look in drop-down, then press Enter or click Open.
- Double-click on the file name in the scrolled list under the Look in drop-down.
- Type the name of the file in the File name text field, then press Enter or click Open. (The insertion point is initially set to highlight the entire text field, so all you have to do is type in the file name.)

To change the format in which the files and folders are listed in the field below the Look in drop-down menu, toggle between the List and Details buttons. These buttons are located in the upper right-hand corner of the dialog.

### 2.2.7. Basic Operations

The basic operation of Tecplot controls will be familiar to anyone who has used Motif or Windows interfaces. Most actions are performed by clicking the mouse, that is, pressing and releasing the left mouse button. (If your mouse is configured for left-hand use, then the word "click" means depress and release the right mouse button.)

Another common mouse action is dragging, which is performed by pressing the left mouse button, then, without releasing the button, moving the pointer. Dragging is used in resizing frames, creating and modifying geometries, and to alter or adjust data.

Clicking and dragging can be combined with keyboard actions to produce different actions. Tecplot makes extensive use of the Ctrl-click (clicking the mouse while holding down the Ctrl key) in its probing feature. See Chapter 26, "Probing," for details. Similarly, in lists which permit multiple selections, you select a single item by clicking on it. You select a range of items by clicking on the item at one end of the range, then Shift-clicking (clicking the mouse while holding down the Shift key) on the item at the other end of the range (you can also simply drag the mouse from the first selection to the last). You select an arbitrary set of items by clicking on the first item, then Ctrl-clicking on subsequent items until all desired items are selected.

The primary tasks done with the mouse are to select objects and options, and choose actions. To select an object means different things for different types of objects:

- For check boxes and option buttons, to select means to click on the desired option. A selected check box or option button is either filled or marked with an X, as shown in Figure 2-13, while an unselected check box or option button is empty.

- For list box items and objects in the Tecplot workspace, including frames, text, geometries, zones, and so on, to select means to highlight and make the object the recipient of subsequent actions. For example, before you can make any changes in any of the Plot Attributes dialogs, you must select one or more zones (field plots) or XY-mappings (XY-plots).

To choose an action means to click on a button or menu item that performs some specified action. For example, if you click Close on a dialog, the dialog closes. If you click New Layout in the File menu, Tecplot clears the workspace and creates a new, empty frame.

The terms click, select, and choose are sometimes used interchangeably. It is useful, however, to keep in mind that select in general means to "select an item to operate on," while choose in general means to "pick an action."

To select an object in the workspace, simply click on it. To select an object and call up the dialog used to modify the object, double-click on the object. For example, if you double-click on a piece of text, the Text dialog appears so that you can edit or reformat the text. You can select the object and click on the Details button in the sidebar for the same effect. You can select groups of items, then act on them all at once. To select a group of items, perform the following steps:

1. On the sidebar, select  .
2. In the workspace, click-and-drag the pointer. A rubber band box appears.
3. Drag the pointer until all the desired items are enclosed in the rubber band box, as shown in Figure 2-20.
4. Release the mouse button. The Group Select dialog appears, as shown in Figure 2-21.
5. Select the objects you want to select using the appropriate check boxes.
6. Click OK to select the desired items.

An alternative way to select multiple objects is to hold the Shift key down and click on one object at a time.

## 2.2.8. Positioning and Resizing Objects

Selected objects such as frames, text, geometries, legends, and so forth, may be moved either by clicking and dragging, or by using the arrow keys on your keyboard. Arrow keys move objects in one pixel increments. For more information on moving and resizing frames, see Section 3.1.3, "Sizing and Positioning Frames."

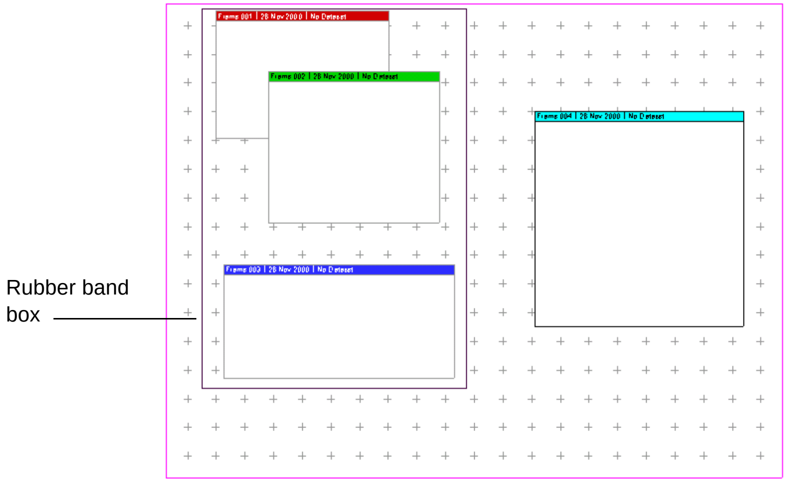



**Figure 2-20.** A rubber band box around three frames.

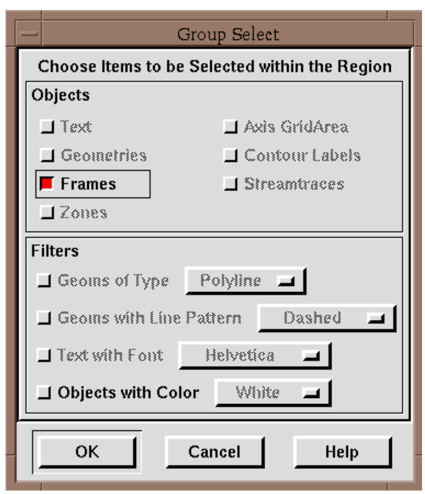



**Figure 2-21.** The Group Select dialog.

To scale selected objects proportionally, maintaining the vertical to horizontal aspect ratio, select the object, then press “+” on your keyboard to enlarge or “-” to reduce. Double-clicking a selected object will bring up its property attributes dialog. For example, if you doubled-clicked on a geometry, the Geometry dialog would appear.

### 2.2.9. The Quick Edit Dialog

Those aspects of the plot that affect how the individual layers are drawn are called plot attributes. You control these attributes using the options under the Field menu (for 2- and 3-D plots) or the XY menu (for XY-plots). You can also control many of these attributes using the Quick Edit dialog, shown in Figure 2-22.

To use the Quick Edit dialog, select one or more objects in the workspace, then click the appropriate button to change the attribute of the selected object(s).

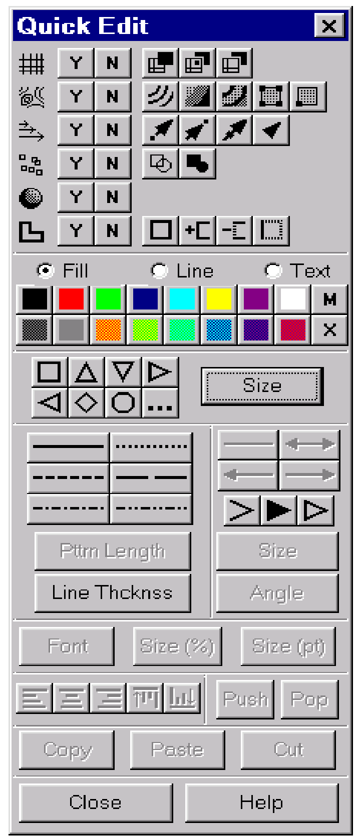



**Figure 2-22.** The Quick Edit dialog.

## 2.3. Help

Tecplot features an extensive Help system, which is fully integrated into the Tecplot interface. Quick help on menu items and sidebar controls is available from the status line, while detailed help is accessible in any of the following ways:

- Press the F1 key anywhere in the Tecplot window. If the pointer is over the sidebar, Quick Edit dialog, or a menu, the F1 key provides context-sensitive help on that control or menu. Otherwise, F1 calls up the Contents page of Help via your Web browser.
- Select Contents from the Help menu. This calls up the Contents page of the Tecplot help file via your Web browser.
- Click Help on any dialog.

Figure 2-23 shows Tecplot's Help as it appears in a Web browser in Windows. It supports text search, has many hypertext links, and provides detailed information on all menus and dialogs.

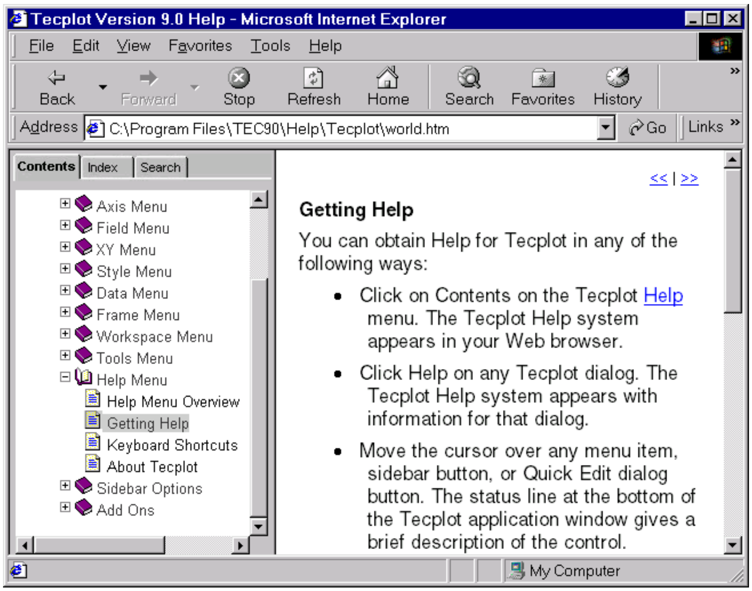



**Figure 2-23.** Tecplot Help in a Windows Web browser.

Your answer may be in *Technical Support Notes* at www.amtec.com/support. Help is also available from 6:30 a.m. to 5:00 p.m. Pacific Standard Time from Tecplot Technical Support at **425.653.9393**. (Be sure to ask for Tecplot Technical Support.) You may also send e-mail to **support@amtec.com** with your questions.

# CHAPTER 3 Frames and the Workspace

No matter which type of plot you want to create, certain operations occur repeatedly within Tecplot. Those operations concerning files are covered in Chapter 4, “Data Organization.” Operations concerning software are covered here. These options are discussed:

- **Working with frames**: All plots are created in a plotting frame—a boxed area in the workspace that acts like a sub-window. You control each frame format individually.
- **Managing your workspace**: The workspace and paper controls determine the color and orientation of your paper, as well as the Ruler and Grid, which help you precisely size and position objects.
- **Understanding coordinate systems**: It is important to understand when and where Tecplot uses a number of different coordinate systems.
- **Controlling the plot view**: Zooming, translating, and fitting your plot in a frame.
- **Copying, cutting, and pasting**: Many plot elements may be cut or copied from the workspace and pasted back into other plot elements.

## 3.1. Working with Frames

All of Tecplot’s plots and sketches are drawn inside frames. By default, the Tecplot window contains one frame, but you may add additional frames, up to a total of 128. You may resize and reposition frames, modify their background color, and specify whether their borders and headers appear. Tecplot acts upon only one frame at any given time. This is the current frame.

### 3.1.1. Creating Frames

You create new frames interactively, by drawing them in the workspace. If you will be printing your plots, you should draw frames within the paper displayed in the workspace. However, this is not required.

To create a new frame:

1. From the sidebar, select , or choose Create from the Frame menu.
2. Move the pointer into the workspace. The pointer becomes a cross-hair.
3. Move the cross-hair to the desired location of one corner of the frame, then click the left mouse button and drag. A rubber band box shows the outline of the frame.
4. When the rubber band box is the desired size and shape, release the mouse button.

### 3.1.2. Deleting Frames

You can delete frames one at a time using the Delete Current Frame option under the Frame menu, or delete frames singly or in groups using the Clear option under the Edit menu.

To delete a single frame:

1. In the workspace, click anywhere in the frame to make it the current frame.
2. From the Frame menu, choose Delete Current Frame. Or, if the frame is selected (by clicking on its border or header) you can choose Cut or Clear under the Edit menu.

To delete a group of frames:

1. Select the group of frames as described in Section 2.2.7, “Basic Operations.” The Group Select dialog will appear.
2. In the Objects region, deselect all check boxes except Frames. (If your rubber band box encloses any frames, the Frames check box will be selected for you automatically. Otherwise, the Frames check box will be desensitized.) All the frames within the rubber band box are selected.
3. From the Edit menu, choose Clear, or, with the keyboard focus in the Tecplot window, type Delete. A warning dialog appears asking if you really want to delete the selected items.
4. Click OK to delete the selected frames; click Cancel to retain the selected frames.

### 3.1.3. Sizing and Positioning Frames

You can size and position frames in four ways: with your mouse, with the arrow keys on your keyboard, specifying exact coordinates using the Edit Current Frame dialog, or from the Frame menu, choosing Fit all Frames to Paper.

**3.1.3.1. Sizing and Positioning Frames Using the Mouse.** If you click anywhere on the frame header or frame border, resize handles appear at the corners and midpoints of the frame. Drag any of these resize handles to resize the frame. The resize handles on the top and bottom of the frame allow resizing only vertically; the resize handles on the left and right of the frame allow resizing only horizontally. The resize handles on the four corners allow simultaneous resizing vertically and horizontally. You can also obtain the resize handles by selecting a group of frames, as described in Section 2.2.7, “Basic Operations.”

To scale the frame or frames proportionally, maintaining the vertical to horizontal aspect ratio, select the frames so that the resize handles appear. Press “+” on your keyboard to enlarge the frames, “-” to reduce them.

**3.1.3.2. Positioning Using the Arrow Keys.** If you click anywhere on the frame header or frame border, handles appear at the corners and midpoints of the frame. Using the arrow keys on your keyboard, you can move the frame up, down, left or right in one-pixel increments for precise locating. You cannot resize a frame using arrow keys.

**3.1.3.3. Sizing and Positioning Frames Using the Edit Current Frame Dialog.** If you want precise control over the size of your frames and where they are located, use the Edit Current Frame dialog to specify the exact location for the frame’s left and top sides, together with the frame’s width and height. You use the same units in this dialog as are currently displayed in the workspace rulers, which can be shown as inches or centimeters.

To precisely position and size your frame:

1. From the Frame menu, choose Edit Current Frame. The Edit Current Frame dialog appears as shown in Figure 3-1. (You can also get to this dialog by double-clicking on the header or border of the current frame.)
2. Enter the position of the left side of the frame in the Left Side text field and enter the position of the top side of the frame in the Top Side text field, using Paper Ruler units.
3. Enter the width and height of the frame, using Paper Ruler units, in the Width and Height text fields. Units other than Paper Ruler may be specified by typing them after the number. For example, cm for centimeters, in for inches, or pix for pixels.
4. Click Close to close the Edit Current Frame dialog.

### 3.1.4. Modifying the Frame Background Color

You can alter the frame background color for a variety of effects. To create a transparent frame, turn off the background color completely. Use transparent frames to create overlay plots showing contour lines for two or more contour variables.

To turn off the background color and create a transparent frame:

1. From the Frame menu, choose Edit Current Frame. The Edit Current Frame dialog appears.

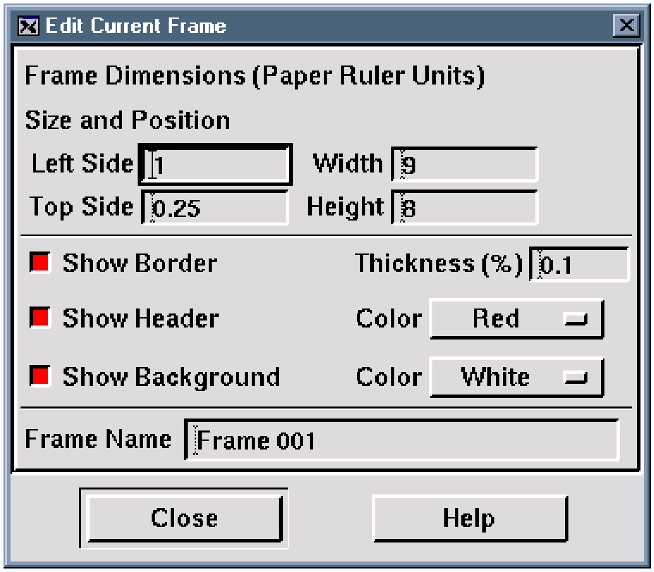



**Figure 3-1.** The Edit Current Frame dialog.

2. Deselect the Show Background check box. (By default, this check box is selected.)
3. Click Close.
4. Click Redraw All in the Tecplot sidebar to redraw the transparent frame and any frames lying beneath it.

To choose a different background color:

1. From the Frame menu, choose Edit Current Frame. The Edit Current Frame dialog appears.
2. Verify that the Show Background check box is selected. (By default, this check box is selected.) Immediately to the right of the Show Background check box is a drop-down labeled Color containing Tecplot's basic colors.
3. From the Color menu, select the desired background color.
4. Click Close.
5. Click Redraw to redraw your frame with the new background color.

### 3.1.5. Controlling Frame Borders and Headers

Every frame is surrounded by a border and is topped with a header. The frame border acts much like a picture frame, giving a clear visual outline of the drawing region. Sometimes, however, the picture frame may be undesirable. For example, if you are building a composite plot out of multiple frames, the frame borders may detract from the appearance of the finished plot.

Tecplot allows you to turn off frame borders for any frame. You can make the frame border invisible, in which case the frame header is also invisible, or you can display the border but not the header. You can also control the thickness of the frame border. You specify the thickness in frame units, as a percentage of the frame height.

The frame header contains information that can be configured by the user, and defaults to

```
"&(FrameName) | &(date) | &(DataSetTitle)"
```

This displays information about the name of the frame, the date the frame was created or revised, and, if applicable, the title of the current data set (these defaults can be changed in your configuration file; see the **`$!GLOBALFRAME`** command in the *Tecplot Reference Manual*).

The frame header is displayed only when both the Show Border and Show Header check boxes are selected in the Edit Current Frame dialog. By default, both check boxes are selected. However, if you turn off the frame border by deselecting the Show Border check box, the header will be turned off as well. You can choose any of Tecplot's basic colors for the frame header.

On most screens, the header information is difficult to read unless you are zoomed into the paper (for example, by selecting Fit All Frames from the Workspace menu).

To modify the display of frame borders and headers:

1. From the Frame menu, choose Edit Current Frame. The Edit Current Frame dialog appears.
2. Set your header and border settings as desired.
3. Click Redraw.

Positioning and resizing borderless frames can be somewhat frustrating, because it is difficult to click on boundaries that cannot be seen. For this reason, Tecplot by default displays a dashed representation of the "invisible" frame borders. These dashed lines do not appear in graphics formatted for printing, nor in EPS files, but they do appear in bitmap files created from Tecplot. You can turn off this feature by toggling the option Show Invisible Borders in the Frame menu.

When the Show Invisible Borders option is on (the default), a small box (or check mark in Windows) appears to the left of the words "Show Invisible Borders" in the Frame menu. When the option is off, no box appears in the menu.

To turn off display of invisible borders:

1. From the Frame menu, click Show Invisible Borders while the small box (or check mark) appears to the left of the words "Show Invisible Borders."

### 3.1.6. Modifying the Frame Name

You may alter any frame's name so that it reflects the contents of the frame. One advantage of giving frames meaningful names is that they can be easily distinguished in the Order Frames dialog. See Section 3.1.7, "Pushing and Popping Frames."

To change the name of the current frame:

1. From the Frame menu, choose Edit Current Frame. The Edit Current Frame dialog appears.
2. Change the frame name to the desired value.
3. Click Close.

### 3.1.7. Pushing and Popping Frames

If you have overlapping or overlaid frames, there will be times when you want to expose, or pop, underlying frames. For frames that are partially exposed, you can do this by clicking on the exposed portion of the frame (in any mouse mode except Create Frame). For frames that are completely obscured, you pop underlying frames by pushing the covering frames to the back of the plot, or use the Order Frames dialog found under the Frame menu.

To push a frame to the back of the plot:

1. In the Tecplot workspace, click on the frame to make it the current frame.
2. From the Frame menu, choose Push Current Frame Back.

If you have multiple overlaid frames, you may need to do the above steps repeatedly until the desired frame is on top, or pop a specific frame by name using the Order Frames dialog.

If part of a frame is visible, you can pop it to the top of the view stack by clicking on it. Alternatively, you can pop a frame by name using the Order Frames dialog:

1. From the Frame menu, choose Order Frames. The Order Frames dialog appears, as shown in Figure 3-2.
2. Select the desired frame by name from the list and click Pop. Alternately, double clicking a frame name in the list will select and pop the frame.
3. To change the display order of the frame names within the list, select the List By option menu. When listed by name the frame names are displayed within the list alphabetically. When listed by draw order, the frames are displayed within the list in the order that they are drawn.
4. Click Close.

**Note**: The List By option does not affect the actual frame order within the workspace, only the display of the names within the list.

To change a frame's designated name, see Section 3.1.6, "Modifying the Frame Name."

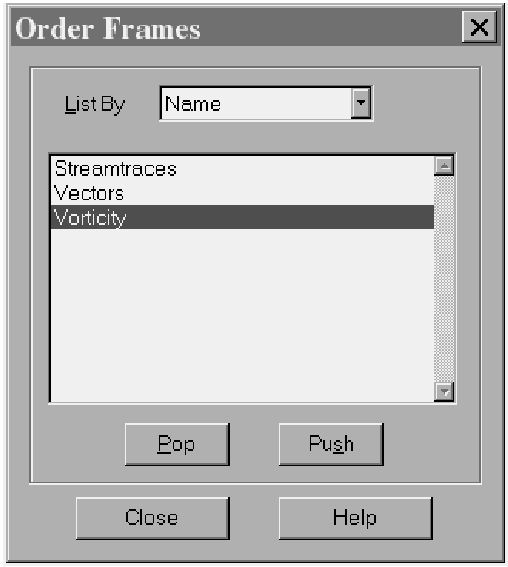



**Figure 3-2.** The Order Frames dialog, accessed via the Frame drop-down menu. Frames have been named using the Edit Current Frame dialog under the Frame drop-down menu.

## 3.2. Managing Your Workspace

The workspace is the entire region in which you can create Tecplot frames, including, but not limited to, the region covered by the Tecplot paper. You may find yourself using only the paper region in creating the screen plots. This is a natural way to work, but not essential, since the paper only limits the printing of the plots. If you are creating plots for screen use only, you may find it useful to use the entire workspace. One way to do this is simply to turn off display of the Tecplot paper.

### 3.2.1. Setting Up the Tecplot Paper

Tecplot's representation of paper in the workspace allows you to lay out your plots in precisely the way you would like them to be printed. If you place a frame somewhere on this paper, then print the resulting plot, the frame will appear in the exact relative location on the printed paper.

Tecplot gives you tremendous flexibility and control in setting up your paper. You can control the size, orientation, and even the color of your paper. You can also turn off the screen representation of the paper.

**3.2.1.1. Controlling Paper Size.** Tecplot offers the following six paper sizes:

- **Letter:** Standard U.S. letter size, 8 1/2 x 11 inches.
- **Double:** Standard U.S. ledger size, 11 x 17 inches.
- **A4:** Standard European letter size, 21 x 29.7 centimeters.
- **A3:** Standard European size, 29.7 x 42 centimeters.
- **Custom 1:** Default is 8.5 x 14 inches.
- **Custom 2:** Default is 8 x 10 inches.

To choose a paper size:

1. From the File menu, choose Paper Setup. The Paper Setup dialog appears.
2. In the Size region of the Paper Setup dialog, select the desired paper size.

In Windows, you can also set the paper size in the Print dialog under the File menu.

All of Tecplot's paper sizes can be customized using options in Tecplot configuration or macro files. We recommend that you only change the dimensions of the Custom 1 and Custom 2 paper sizes.

**3.2.1.2. Controlling Paper Orientation.** Tecplot layouts can be created as either landscape or portrait plots. In landscape orientation, the long axis of the paper is horizontal, while in portrait orientation the long axis is vertical. Portrait orientation uses the specified paper's width for the horizontal dimension, while landscape uses it for the vertical dimension. You specify the orientation as part of paper setup. The default is landscape.

To specify a paper orientation:

1. From the File menu, choose Paper Setup. The Paper Setup dialog appears.
2. In the Orientation region of the Paper Setup dialog, select the desired orientation.

**3.2.1.3. Turning Off the Screen Paper.** If you are creating plots simply for display on your screen, you need not be constrained by the limits of the printed page. You can turn off the screen representation of the paper and more freely use the full workspace.

To turn off the screen paper:

1. From the File menu, choose Paper Setup. The Paper Setup dialog appears.
2. Deselect the check box labeled Show Paper on Screen.

**3.2.1.4. Controlling the Paper Color.** You can set up your paper to show any of Tecplot's basic colors as a background color (called the "paper fill color") on your screen, and also allow Tecplot to use that color when printing to a color printer.

To specify the paper fill color:

1. From the File menu, choose Paper Setup. The Paper Setup dialog appears.
2. From the drop-down labeled Paper Fill Color, choose the desired color from the list of Tecplot's basic colors.

When you are printing, you can have Tecplot flood the paper with your specified paper fill color. By default, the paper fill color is ignored during printing.

To use the paper fill color when printing:

1. From the File menu, choose Paper Setup. The Paper Setup dialog appears.
2. Select the check box labeled Use Paper Fill Color when Printing.

## 3.2.2. Setting Up Grids and Rulers

The workspace grid provides a convenient guide for placing objects on your Tecplot paper. When placing text or geometric shapes, you can choose to snap the anchor points of the shapes to the grid.

Rulers help you size objects such as frames, text, and geometries by providing a reference length. You can visually gauge the relative size of objects on your screen by comparing them to the vertical and horizontal rulers. Tecplot allows you to draw the rulers in centimeters (cm), inches (in), or points (pt), or not draw them at all.

**3.2.2.1. Controlling the Workspace Grid.** Tecplot allows you to select the grid spacing from several pre-set sizes in dimensions of centimeters (cm), inches (in), or points (pt). You can also specify not to show the grid. By default, the grid is shown. The grid is not shown if the paper is not visible onscreen, or if the Show Grid check box is deselected.

To turn off the grid:

1. From the Workspace menu, select Ruler/Grid. The Ruler/Grid dialog appears.
2. Deselect the Show Grid check box.

To specify the grid spacing:

1. From the Workspace menu, select Ruler/Grid. The Ruler/Grid dialog appears.
2. From the drop-down titled Grid Spacing, choose the desired spacing.

**3.2.2.2. Controlling the Workspace Ruler.** Tecplot allows you to select the ruler markings from several pre-set sizes in dimensions of centimeters (cm), inches (in), or points (pt). You can also specify whether to show the ruler. By default, the ruler is shown. The ruler is not shown if the Show Ruler check box is deselected.

To turn off the ruler:

1. From the Workspace menu, select Ruler/Grid. The Ruler/Grid dialog appears.
2. Deselect the Show Ruler check box.

To specify the ruler spacing:

1. From the Workspace menu, select Ruler/Grid. The Ruler/Grid dialog appears.
2. From the drop-down titled Ruler Spacing, choose the desired spacing.

### 3.2.3. Maximizing Your Workspace

You can create plots up to the full size of the workspace, and you can force that workspace to fill the Tecplot window, hiding the sidebar and menu bar.

To maximize your workspace:

1. From the Workspace menu, choose Maximize Workarea. The sidebar and menu bar disappear.
2. To return the sidebar and menu bar, simply click anywhere in the maximized workspace.

## 3.3. Coordinate Systems

Tecplot manages a number of coordinate systems. Four of these coordinate systems that are important to know about are the paper, frame, 2- and 3-D physical coordinate systems.

Figure 3-3 shows the origins of each coordinate system and how each coordinate system relates to the other. Note that only one of the 2- or 3-D physical coordinate systems is in effect at any given time, depending on the frame mode.

The 2- and 3-D physical coordinate systems are the coordinate systems in which the X-,Y-, and or Z-coordinates of your data points are plotted. The 2-D physical coordinates are often referred to as grid coordinates.

The frame coordinate system is shown in Figure 3-4. The vertical axis of this coordinate system always runs from zero at the bottom border line of the frame to 100 at the top of the frame. The horizontal axis likewise runs from zero to 100 from left to right along the bottom edge of the frame. The distance of one horizontal unit is not necessarily equal to the distance of one unit in the vertical direction, since frames take on almost any aspect ratio.

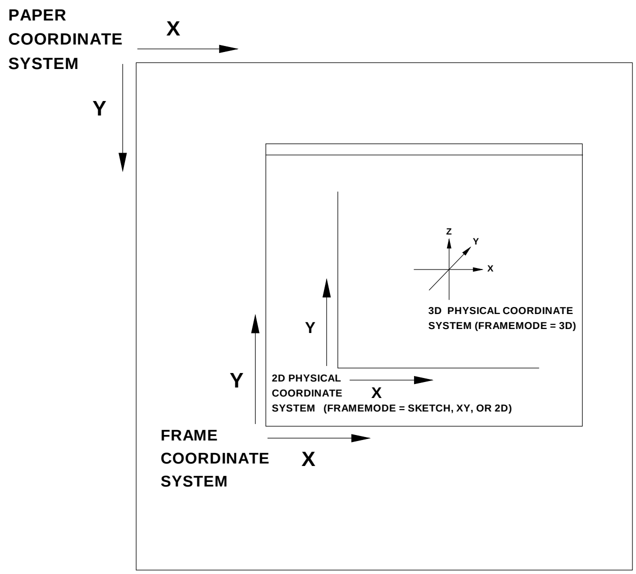



**Figure 3-3.** Coordinate systems in Tecplot.

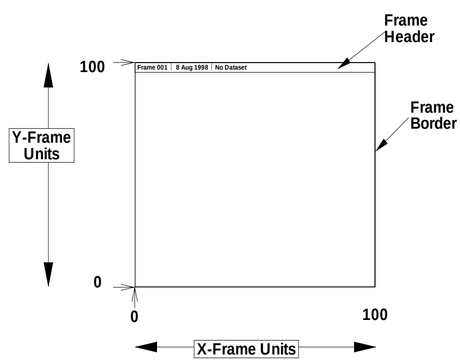



**Figure 3-4.** The frame coordinate system.

Tecplot uses the height of the frame for objects that are scaled by frame units (such as font size). Whenever you enter a frame unit value into a Tecplot dialog, or when you are setting frame size and position on your paper, you may specify a different unit system as a suffix. Tecplot converts the value to frame units (or paper units when sizing or positioning a frame) for you. The valid suffixes are “in” (inches), “pt” (points), “cm” (centimeters), and “pix” (pixels). For example, if you want a piece of text exactly one inch away from the left edge of your frame you can enter “1in” in the X-Origin field. Tecplot converts the value to the appropriate frame unit value.

## 3.4. Modifying Your View

There are two types of views inside Tecplot. The first is the view of your data inside a frame. Each frame can have a separate view of its data. This view is controlled by the View menu.

The second type of view is the view of the frames and paper inside the workspace. This view is controlled by options in the Workspace menu.

Both types of views may be controlled by the two view mouse modes in the sidebar: Zoom and Translate. However, other actions differ depending on whether you are acting on a frame or the workspace. These are covered in the following subsections.

### 3.4.1. Modifying the View of Your Data within a Frame

The view of your Tecplot data is the position, size, and orientation of the plot within the Tecplot frame. The View menu contains controls to help you adjust the view to your tastes, and also to copy the view from one frame to another. The View menu contains the following options:

- **Redraw:** Redraws the current frame.
- **Zoom:** Turns on the Zoom mode, which you use to zoom into the current frame.
- **Fit to Full Size:** Resizes the plots so that all data points, text, and geometries are included in the frame. This is the initial view for XY and 2D frame modes.
- **Data Fit:** Resizes the plot so that all the data points are included in the frame. Text and geometries are not considered.
- **Center:** Centers the plot in the frame.
- **Translate/Magnify:** Turns on the Translate mode and calls up the Translate/Magnify dialog, which you use to move and resize your plot with respect to the frame.
- **Last:** Restores the previous view.
- **3D Rotate:** Calls up the 3D Rotate dialog, which you use to rotate 3-D images. For further information, see Section 9.6.1, “3-D Rotation.”

- **3D View Details:** Calls up the 3D View Details dialog, which sets the view position and angle for 3-D images. For further information see Section 9.6.1, “3-D Rotation.”
- **Copy View:** Copies the current frame view to a buffer, it can then be pasted onto another frame.
- **Paste View:** Pastes a copied view onto the current frame.

We have already used Redraw; it is on the View menu to remind you of the keyboard shortcut Ctrl-R to redraw your plot. The 3D Rotate and 3D View Details options are only active when the current frame is in 3D frame mode.

This section discusses the remaining View options. Shortcuts are provided for most of these controls. For Zoom, you can use the  button in the sidebar to select the Zoom mode. For Translate/Magnify, you can use the  button in the sidebar. For Fit to Full Size, you can use the keyboard shortcut Ctrl-F almost anytime the Tecplot application window has the input focus. For Last, you can use the keyboard shortcut Ctrl-L. For Paste View, you can use the keyboard shortcut Ctrl-A. There are no shortcuts for Data Fit, Center, and Copy View.

**3.4.1.1. Smooth Zooming and Translation with Your Mouse.** The middle and right mouse buttons allow you to smoothly zoom and translate your data. Your middle mouse button (or Ctrl-right click) zooms smoothly, and your right mouse button translates data. (See Appendix C, “Mouse and Keyboard Operations,” for additional mouse functionality.) This advanced functionality is available in:

- **All Contour Modes.**
- **Streamtrace Placement.**
- **Slicing.**
- **All 3-D Rotation Modes.**
- **All Geometry Modes (Except Polyline).**
- **Zooming.**
- **Translate/Magnify.**
- **Probing.**
- **Zone Creation.**

**3.4.1.2. Zooming into Your Data within a Frame.** You can use Tecplot’s Zoom feature to quickly zoom into a portion of your plot. To zoom into your plot, select the Zoom tool from the sidebar. The pointer becomes a magnifying glass. Drag to draw a box around the region you want to magnify. Your plot is resized to fit the longest dimension of the zoom box you created on the screen. You can quickly zoom into a region by positioning the magnifying glass

and clicking your left mouse button. This magnifies the region by 200 percent and centers the zoom on the position of the magnifying glass.

For certain types of data, it may be useful to zoom into successively smaller regions until the area of interest is revealed in adequate detail. For example, look at the sample finite-element data file `feexchng.plt`. At full size, you see the whole data set, and see that there is an area of interest containing many circles. You zoom in on the circles, and then zoom in again on a single circle. Finally, you zoom in on the boundary of a circle. This sequence is illustrated in Figure 3-5. At any stage of the zoom, you can use Ctrl-L to return to the previous (last) view, or Ctrl-F to return to the full size initial view.

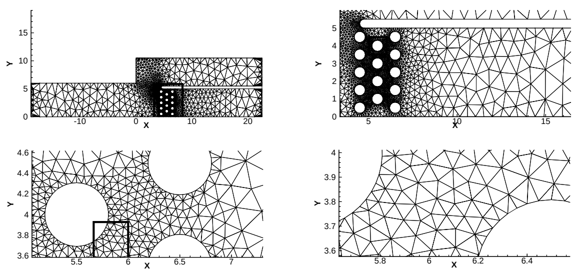


**Figure 3-5.** Zooming into a plot.

You can fit one or all frames to the workspace by using the Fit Selected Frame or Fit All Frames option under the Workspace menu. Both options are alternative methods of zooming the paper.

To return to the default paper view, choose Fit Paper from the Workspace menu.

**3.4.1.3. Translating and Scaling Your Data within a Frame.** You can use Tecplot’s Translate/Magnify feature to translate the view of your plotted data. Translating moves the image of your data with respect to the current frame. You can translate plots in any direction within the frame. The Translate/Magnify feature is available as both a sidebar tool and as a dialog, shown below.

The following options and shortcuts are available for the Translate/Magnify dialog:

**Up, Down, Left, Right:** Use the four arrow buttons to translate your image in the desired direction.

**Magnification Factor:** Increase or decrease the magnification by using the arrow buttons or by entering a value in the text field.

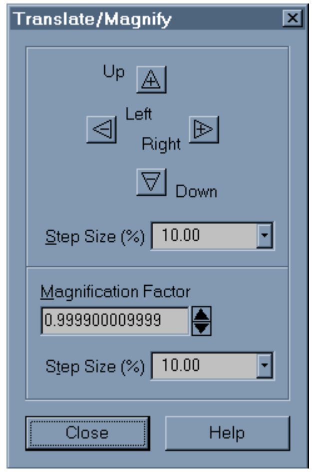



**Figure 3-6.** The Translate/Magnify dialog.

**Step Size (%):** Control the step size for each arrow button using the pre-set ranges in the drop-down, or by entering your own value in the text field.

To translate your image using Translate/Magnify tool from the sidebar:

1. On the sidebar, click . The pointer changes to an all-direction cursor.
2. In the workspace, drag in the direction you want to transfer the image.
3. While translating in 2D frame mode, you see a trace of your data. That is, you see a wire frame sketch of what the plot will look like when you are done. Redraw your plot when you are done to see the entire plot. The number of lines allowed in the trace can be changed through the Performance button on the sidebar.

Translate/Magnify tool mode offers the following keyboard options:

**+:** Increase scale of images.

**- :** Decrease scale of images.

**3.4.1.4. Fitting Your Data to a Frame.** You can use the Fit to Full Size option from the View menu to restore the initial view of your data after extensive zooming, scaling, or translating. Tecplot performs the Fit to Full Size operation when it first displays your data set. You can perform the operation in either of the following ways:

- Select Fit to Full Size from the View menu.
- While the window is active, press Ctrl-F.

**3.4.1.5. Restoring the Last View of Your Data within a Frame.** Tecplot does not have an "Undo" option. Any data alterations are permanent, but you can step backward through the resizings and repositionings of your plot.

Any time you change the view of a frame, either by zooming, centering, translating, or fitting the plot, the previous view is placed in a view stack. Each frame has four view stacks, one for each frame mode. Each view stack stores the last 16 views for that frame mode. You can move back through the view stack by choosing Last from the View menu repeatedly, or more quickly by typing Ctrl-L repeatedly.

**3.4.1.6. Copying and Pasting Views between Frames.** When you are working with multiple frames attached to the same data set, it is often useful to make your view changes to one frame, and then propagate those changes to the other frames. You can do this using the Copy View and Paste View options under the View menu, as follows:

1. Make the changes (zooming, translating) you want to make to one frame.
2. From the View menu, choose Copy View.
3. Click in another frame sharing the same frame mode.
4. From the View menu, choose Paste View. (Or type Ctrl-A.)

**Note:** Copy View and Paste View only affect the ranges of XY-axis and tick mark spacing. For complete duplication, use Copy Style to File and Paste Style from File from the Style menu.

### 3.4.2. Modifying the View of Frames and Paper within the Workspace

The view of your frames and the paper within the workspace is controlled through the Workspace menu. This is called the workspace view. The Workspace menu contains the following options to control the workspace view:

- **Redraw All:** Redraws all frames as well as the paper.
- **Fit Selected Frames:** Resizes the workspace view such that the currently selected frames are included in the view.
- **Fit All Frames:** Resizes the workspace view such that all frames are included in the view.
- **Fit Paper:** Resizes the workspace view such that the paper fits into the view.
- **Last View:** Restores the workspace to its previous view.

**3.4.2.1. Zooming the Workspace.** To zoom the workspace, use the  sidebar tool. The mouse pointer changes to a magnifying glass. Shift-drag the magnifying glass cursor to draw a box about the region that you want to magnify. The plot is resized such that the longest dimension of the zoom box fits into the workspace.

**3.4.2.2. Translating and Scaling the Workspace.** You can use the  sidebar tool to translate and magnify the paper and the image simultaneously. Magnifying your on-screen image will not affect the printout size.

To translate the entire paper and image:

1. Click the  sidebar tool to enter Translate/Magnify mode.
2. Shift-click to operate on paper and image simultaneously. (Leave the mouse button down.)
3. Drag to move the paper.

To magnify the entire paper and image (operates on the on-screen paper and image only):

1. Click the  sidebar tool to enter Translate/Magnify mode.
2. Shift-click to operate on paper and image simultaneously. Remember to leave the mouse button down.
3. Press “+” to magnify paper and image, “-” to reduce.

You can move and rescale the paper simultaneously so long as you have the mouse button depressed. If you release the mouse button, “+” and “-” will revert to resizing the image of the data.

**3.4.2.3. Fitting the Workspace View.** You can use the Fit Paper option from the Workspace menu to restore the initial view of the paper. Tecplot fits the paper to the workspace when it first starts.

**3.4.2.4. Restoring the Last Workspace View.** You may restore the last workspace view with the Last View option on the Workspace menu.

Any time you change the view of the workspace, the previous view is placed in a view stack, which stores the last 16 workspace views. You can cycle back through the view stack by choosing Last View from the Workspace menu repeatedly.

## 3.5. Copying, Cutting, and Pasting

You can duplicate frames, text, and geometries by copying and pasting using options under the Edit menu (or their keyboard equivalents). You can also cut objects from one location and paste them into another, or throw them away completely. To select all geometries, zones, text or streamtraces in a frame, choose the Select All option from the Edit menu.

**Note:** In Windows, Tecplot’s Cut, Copy, and Paste options work only within Tecplot. However, Copy Plot to Clipboard, under the Edit menu, allows you to copy Tecplot frames and paste

them into other Windows applications. See Section 23.2.3, “Clipboard Capability for Placing Tecplot Images Directly into Other Applications,” for a discussion of this feature.

### 3.5.1. Copying Objects

To create an exact copy of a Tecplot frame, text, or geometry:

1. In the workspace, select the object or objects you want to copy.
2. From the Edit menu, choose Copy, or Ctrl-C on your keyboard. The selected objects are copied to Tecplot’s internal paste buffer.
3. From the Edit menu, choose Paste, or Ctrl-V. A copy of the original object is stacked on top of the original.
4. To place the copy, first select the copy and then move the pointer until it becomes a four-way translation cursor. Drag the copy to the desired location and release the mouse button. (Tecplot will not place an exact replica of a copied text or geometry to the exact same position. In that case, move the original and then paste a copy.)

### 3.5.2. Clearing Objects

To clear an object from the Tecplot workspace means to delete it, without saving it in the Paste buffer. To remove an object and save it in the Paste buffer, use Cut.

To clear an object:

1. In the workspace, select the object or objects you want to clear.
2. From the Edit menu, choose Clear, or Delete on your keyboard. A question dialog appears asking if you are sure you want to delete the selected objects.
3. Click OK.

If you clear the last Tecplot frame, Tecplot automatically creates another frame to replace it.

### 3.5.3. Cutting Objects

To cut an object means to remove it from its current location but store it in the Tecplot Paste buffer.

To cut an object:

1. In the workspace, select the object or objects you want to cut.
2. From the Edit menu, choose Cut, or Shift-Delete or Ctrl-X on your keyboard.
3. You can then paste the object using Paste from the Edit menu.

If you cut the last frame, Tecplot will automatically creates another frame to replace it.

# CHAPTER 4 Data Organization

This chapter describes Tecplot's internal handling and storage of data.

## 4.1. Data Hierarchy

Tecplot structures data in two levels. Figure 4-1 shows the data hierarchy for a simple case.

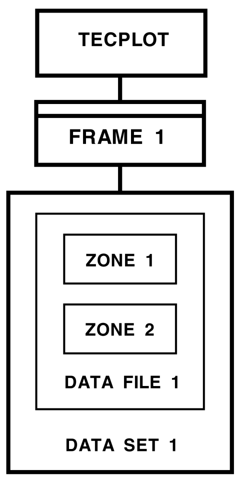



**Figure 4-1.** Tecplot's data set structure.

The highest level is an internal Tecplot data structure known as a data set. A data set consists of one or more zones, blocks of data that make up the whole data set. Zones, the second level in the data hierarchy, can be loaded into a data set from a data file, or created within Tecplot. Starting with a blank frame, a data set is created and assigned to the active frame whenever you read one or more data files into Tecplot, or create a zone within Tecplot. Multiple frames can be attached to the same data set. Figure 4-2 shows how data sets and frames relate to one another.

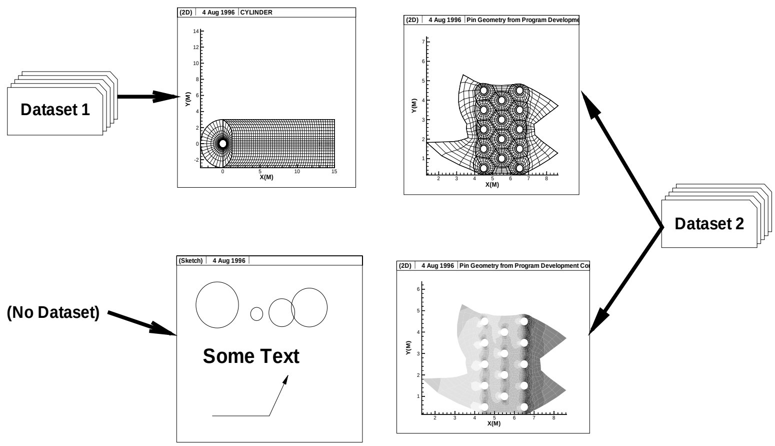



**Figure 4-2.** Data sets and frames.

If more than one data file is read into a frame, Tecplot groups all zones supplied by the files into a single data set. Once in Tecplot, all zones within a data set must contain the same variables defined for each data point. This does not necessarily mean each of your data files needs to have the same number of variables in the same order. See Section 6.1, “Loading Tecplot-Format Data Files,” for instructions on loading dissimilar data files or parts of data files. The number of zones in a concatenated data set will be the sum of the number of zones in the data files that are read. As Figure 4-2 shows, one or more frames can access data from the same data set. Frames using the same data set will initially have the same header color.

Figure 4-3 gives a more complex example of a Tecplot data structure. Frames number 1 and number 2 both access data set number 1, which is made up of one data file containing three zones. Frame number 3 accesses data set number 2 which contains two data files—one with two zones and one with three zones. Frame number 4 uses data set number 3, which contains one data file with two zones.

## 4.2. Multiple Zones

Multiple zones can be used for plotting complex configurations, or subdividing data for convenience in plotting.

You can also represent data that was taken at different time steps, or using measurement methods. The plot on the left in Figure 4-4 provides an illustration of multiple zones. For example, you could use multiple zones to show the measurement of snow depth at several different sta-

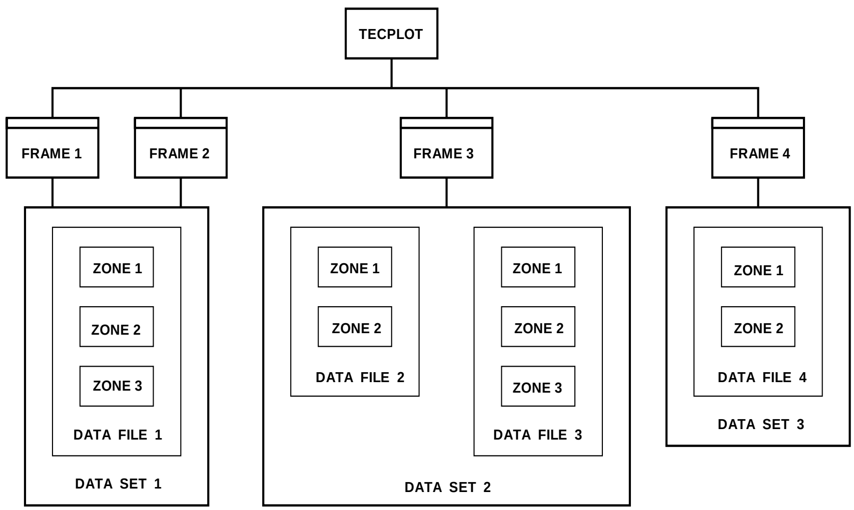



**Figure 4-3.** Example of a complex data structure.

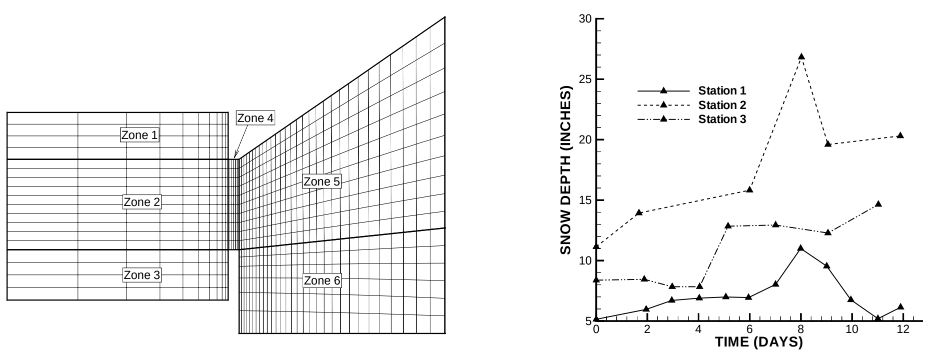



**Figure 4-4.** Example plots showing the use of multiple zones. A 2-D mesh plot is on the right, and an XY-plot is on the left.

tions in an XY-plot. You take measurements once a day at each station, but there are some days on which you cannot get to all of the stations. As a result, when you are finished taking data, you have a different number of data points for each station. Since each set of data has the same number of variables per data point (time and snow depth), you can set up a Tecplot data file with the measurements from each station in a separate zone. In Tecplot, you can define a set of XY-mappings to plot snow depth versus time for any combination of zones (in this example, stations). Figure 4-4 shows a plot of this data.

## 4.3. Data Structuring within a Zone

Tecplot can accommodate two different types of data: ordered and finite-element. The following sections describe each of these two types in detail.

### 4.3.1. Ordered Data

Ordered data is a set of points logically stored in a one-, two-, or three-dimensional array in Tecplot. I, J, and K are used as subscripts to access values within the array. The most common forms for these arrays are:

- **I-ordered:** A one-dimensional array of data points where the I-dimension is greater than or equal to one and the dimension in J- and K-directions is equal to one. The I-dimension thus represents the total number of data points for the zone.
- **IJ-ordered:** A two-dimensional array of data points where both the I- and J-dimensions are greater than one and the K-dimension is equal to one. The number of data points is the product of the I- and J-dimensions.
- **IJK-ordered:** A three-dimensional array of data points where all three of the I-, J-, and K-dimensions are greater than one. The number of data points is the product of the I-, J-, and K-dimensions.

Other ordered data types are also valid but are not typically created in a Tecplot session. These may come from data sets created by other applications wishing to retain a particular data order. They are:

- **J- or K-ordered:** These are the same as I-ordered but the J- or K-dimension is greater than one and the remaining dimensions are equal to one.
- **JK- or IK-ordered:** These are similar to IJ-ordered. In both cases two of the three dimensions are greater than one and the remaining dimension is equal to one.

In general, all discussions in this manual which refer to using I-ordered data may be applied equally to J- or K-ordered data. All three represent a logical one-dimensional array of data. Likewise, all discussions referring to IJ-ordered data may be applied to JK- or IK-ordered data.

**4.3.1.1. I-, J-, or K-Ordered Data Points.** The data points for XY-plots are usually arranged in a one-dimensional array and indexed by one parameter: I for I-ordered, J for J-ordered, or K for K-ordered, with the two remaining index values equal to one. For I-ordered, the most common type, I is as follows: *I=1* at the first data point, *I=2* at the second data point, *I=3* at the third data point, and so forth to *I=IMax* for the last point. At each data point, *N* variables (**V1, V2, ..., V***N*) are defined. If you arrange the data in a table where the values of the variables (*N* values) at a data point are given in a row, and there is one row for each data point, the table would appear something like that shown in Figure 4-5. For example, if you wanted to make a simple XY-plot of pressure versus time, **V1** would be time and **V2** would be pressure.

| | | | | | | | | |
|---|---|---|---|---|---|---|---|---|
| **V1** | **V2** | **V3** | **V4** | **V5** | **V6** | **. . .** | **V***N* | (Values at data point *I* = *1*.) |
| **V1** | **V2** | **V3** | **V4** | **V5** | **V6** | **. . .** | **V***N* | (Values at data point *I* = *2*.) |
| **V1** | **V2** | **V3** | **V4** | **V5** | **V6** | **. . .** | **V***N* | (Values at data point *I* = *3*.) |
| **V1** | **V2** | **V3** | **V4** | **V5** | **V6** | **. . .** | **V***N* | |
| **V1** | **V2** | **V3** | **V4** | **V5** | **V6** | **. . .** | **V***N* | |
| **V1** | **V2** | **V3** | **V4** | **V5** | **V6** | **. . .** | **V***N* | |
| **V1** | **V2** | **V3** | **V4** | **V5** | **V6** | **. . .** | **V***N* | (Values at data point *I* = *IMax*.) |

**Figure 4-5.** Table of values for I-ordered data points (suitable for XY-plots).

You may also input data for 2- and 3-D vector and scatter plots in I-ordered format. You could create a 3-D vector plot by setting the first six variables at each data point to the three physical coordinates (X, Y, Z) and the three velocity vector components (U, V, W). However, if you did this, you would not be able to use features like light source shading, hidden-surface removal, or streamtraces. These features depend upon a mesh structure connecting the data points (see IJ- and IJK-ordering and finite-element surface points in the next sections).

**4.3.1.2. IJ-Ordered Data Points.** The data points for 2- and 3-D surface field plots are usually organized in a two-parameter mesh. Each data point is addressable by a set of the two parameters (I and J) and has four neighboring data points (except at the boundaries). The points are located above, below, to the left, and to the right as shown in Figure 4-6.

At each data point, you would usually define two (or three) spatial variables (X, Y, and Z) plus one or more variables like temperature, velocity components, or concentration. The data points can be plotted in a 2- or 3-D coordinate system where any two (or three) of the variables defined at the data points are the spatial coordinates (by default, the first two or three are used).

A family of I-lines results by connecting all of the points with the same I-index. Likewise, a family of J-lines is formed by connecting all of the points with the same J-index. When both the I-lines and J-lines are plotted in a two-dimensional coordinate system, a 2-D mesh plot

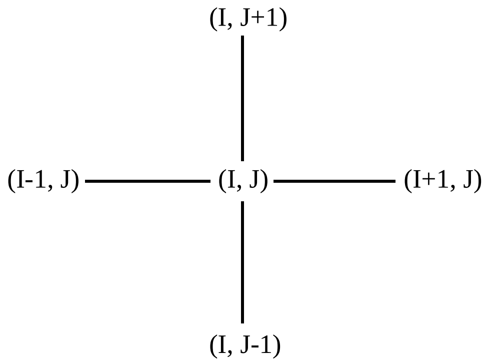



**Figure 4-6.** IJ-ordered data point neighbors.

results as shown in Figure 4-7. When both the I-lines and J-lines are plotted in a three-dimensional coordinate system, a 3-D surface mesh plot results as shown in Figure 4-7.

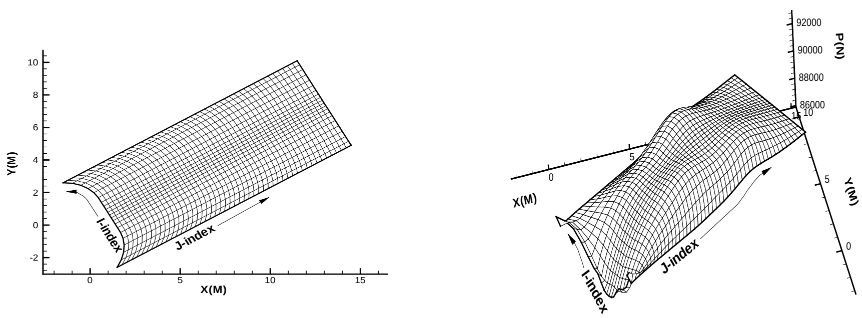



**Figure 4-7.** Left, a 2D frame mode mesh plot of IJ-ordered data points. Right, a 3D frame mode mesh plot of IJ-ordered data points.

The data points lie at the intersections of the I- and J-lines. The points along the I-lines and J-lines need not lie in a straight line. The points may trace out curved, irregularly spaced, and/or nonparallel paths. They may lie in a planar 2-D surface or on a non-planar 3-D surface.

Data organized in IJ-order can also be used for XY-plots. I-order is actually the same as IJ-order with J equal to one. In XY-plots, you can specify the range (maximum and minimum) and skip interval for the I- and J-indices for plotting data points; data points outside of the specified ranges are not plotted. You can also plot the I-lines or the J-lines of an IJ-ordered zone.

**4.3.1.3. IJK-Ordered Data Points.** The data points for 3-D volume field plots are usually organized in a three-parameter mesh. Each point is addressable by a set of three parameters (I, J, and K) and has six neighboring data points (except at the boundaries). These neighbors are located above, below, left, right, in front of, and behind the data point as shown in Figure 4-8.

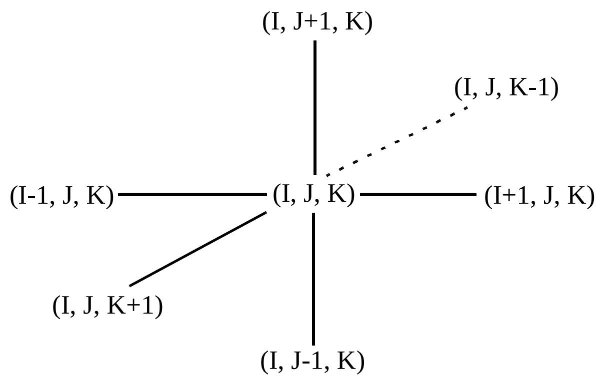



**Figure 4-8.** IJK-ordered data point neighbors.

At each data point, you define three spatial variables (X, Y, Z) plus (typically) one or more variables such as pressure, vector components, and vorticity.

A mesh plot of IJK-ordered data is displayed in Figure 4-9. The directions of the I-, J-, and K-indices are shown. As you can see, the points that define the mesh can form curved, irregularly spaced, and/or nonparallel paths.

**4.3.1.4. I-planes, J-planes, and K-planes.** An important concept in dealing with IJK-ordered data is that of I-planes, J-planes, and K-planes. A K-plane is the connected surface of all points with a constant K-index value. The I- and J-indices range over their entire domains; thus, a K-plane has, in effect, a two parameter ordering, much like IJ-ordering. In fact, IJ-ordered data is identical to IJK-ordered data with the K-index equal to one (*KMax=1*). Note that K-planes are not necessarily planes in the strict sense. They are called K-planes because they exist as planes in logical (IJK) space. In real (XYZ) space, the K-planes may be cones, ellipsoids, or arbitrary surfaces.

An I-plane is the connected surface of all points with a constant I-index value (with *J* and *K* ranging over their entire domains), and a J-plane is the connected surface of all points with a constant J-index value (with *K* and *I* ranging over their entire domains). Figure 4-10, 4-11, and 4-12 show examples of I-, J-, and K-planes.

**4.3.1.5. Plotting IJK-Ordered Data.** Plotting IJK-ordered data is more complex than plotting other ordered data types such as I- or IJ-ordered. With the other data types all data will

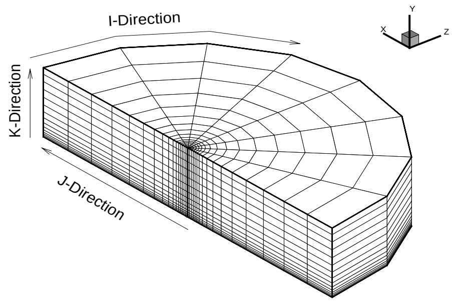



**Figure 4-9.** I-, J-, and K-directions of an IJK-ordered zone.

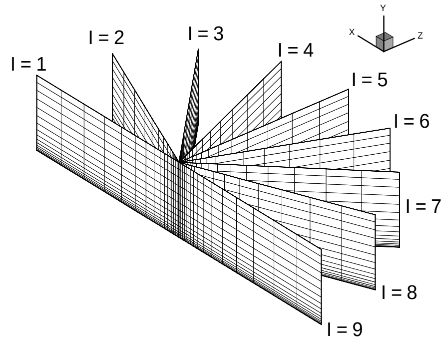



**Figure 4-10.** I-planes of an IJK-ordered zone.

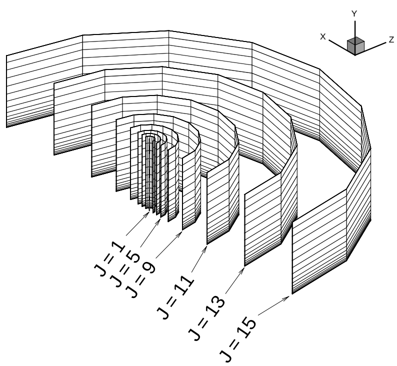



**Figure 4-11.** J-planes of an IJK-ordered zone.

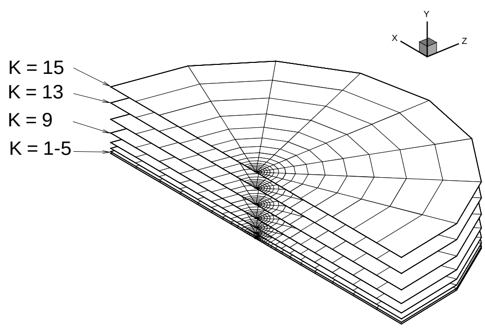



**Figure 4-12.** K-planes of an IJK-ordered zone.

typically be plotted. IJK-ordered data offers more options as to which portions of data will be viewed, especially when creating 2- or 3-D plots. The Volume page of the Plot Attributes dialog allows you to designate which surfaces of IJK-ordered data will be plotted. You may choose to plot just outer surfaces, or you may select combinations of I-, J-, and K-planes to be plotted. For more information see Chapter 20, “Working with 3-D Volume Data.”

### 4.3.2. Finite-Element Data

Finite-element data, also referred to as FE data, is a method of structuring data as a collection of points in 2- or 3-D space with a set of instructions on connecting these points to form elements, or cells.

Finite-element data defines a set of points (nodes) and the connected elements of these points. Finite-element data can be divided into two types:

- **FE-surface:** A set of triangular or quadrilateral elements defining a 2-D field or a 3-D surface.
- **FE-volume:** A set of tetrahedral or brick elements defining a 3-D volume field.

In each of the above data-point orderings there is virtually no limit to the number of data points; the size of your data set is limited only by the amount of physical resources of your computer. You may use a different data point structure for each zone within a data set, as long as the number of variables defined at each data point is the same. Chapter 5, "Formatting ASCII Data for Tecplot," gives detailed information about how to format your data for Tecplot.

**4.3.2.1. Finite-Element Surface Data.** Plotting the connecting lines between finite-element surface data points in a two-dimensional coordinate system results in a mesh like the one shown in Figure 4-15. Plotting a finite-element surface mesh in a 3-D coordinate system results in a mesh like that shown in Figure 4-16.

The values of the variables at each data point (node) are entered in the data file similarly to I-ordered data, where the nodes are numbered with the I-index. This data is followed by another set of data that defines the connections between the nodes. This second section is often referred to as the connectivity list.

You can choose (by zone) to arrange your data in three point (triangle) or four point (quadrilateral) elements. The number of points per node and their arrangement are called the element type of the zone. You may repeat a node in the quadrilateral element type to create a triangle if a mixture of quadrilaterals and triangles is necessary.

**4.3.2.2. Finite-Element Volume Data.** Finite-element volume cells may contain four points (tetrahedron) or eight points (brick). The elements in each zone must be either all tetrahedra or all bricks.

Finite-element volume node connectivity is shown in Figures 4-17 and 4-18. In the brick format, points may be repeated to achieve 4-, 5-, 6-, or 7-point elements. For example, a node list entry of "`n1 n1 n1 n1 n5 n6 n7 n8`" would result in a quadrilateral-based pyramid element. Figure 4-19 shows an example of a finite-element volume mesh plot.

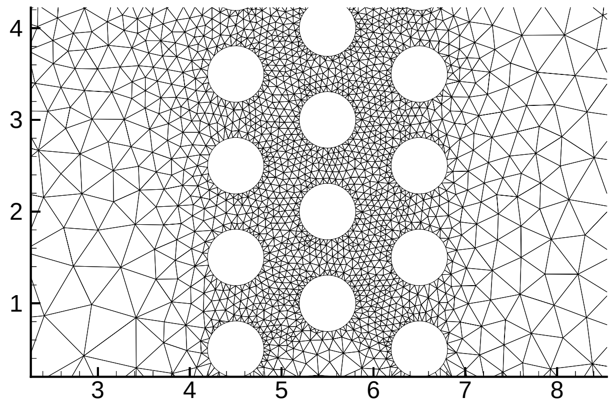



**Figure 4-15.** Mesh plot of finite-element surface data in two dimensions.

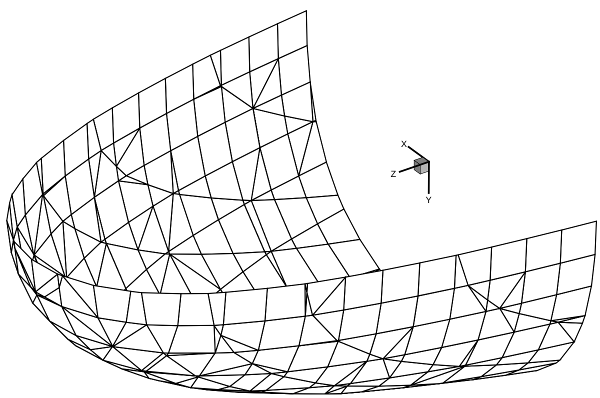



**Figure 4-16.** Mesh plot of finite-element surface data in three dimensions.

In finite-element volume order, the values of the variables at each node (data point) and their connectivity lists are entered in the data file in the same manner as finite-element surface data, as described in Section 4.3.2.1, “Finite-Element Surface Data.”

Finite-element zones of element type brick or tetrahedron are referred to as finite-element volume zones.

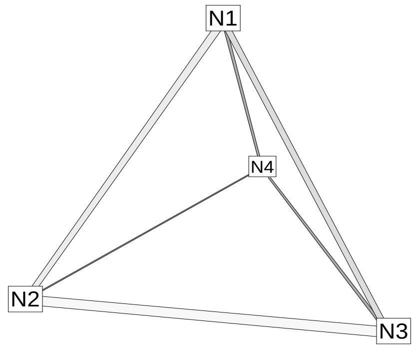



Figure 4-17. Connectivity of tetrahedron FE-volume element.

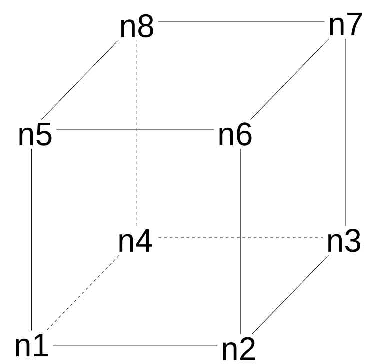



Figure 4-18. Connectivity of brick FE-volume element.

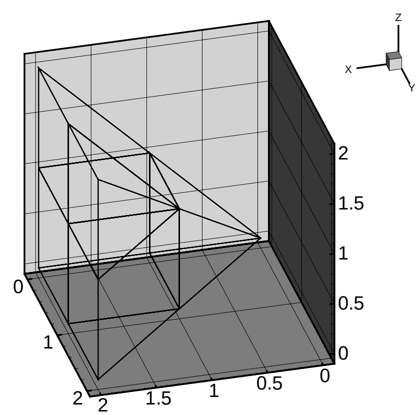
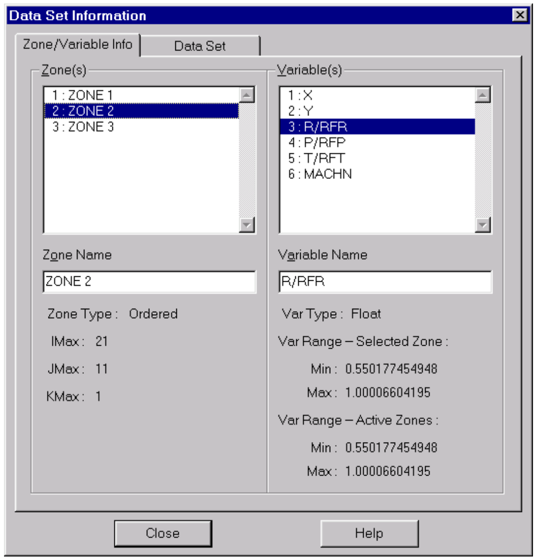



Figure 4-19. An FE-volume mesh plot.

## 4.4. Viewing Data Set Information

The Data Set Information dialog, accessed from the Data Set Info option of the Data menu, gives summary information about the current data set, including the data set title, zone and variable names, and the minimum and maximum values of a selected variable. You can modify the data set title, zone and variable names of any data set. The dialog is shown in Figure 4-20.



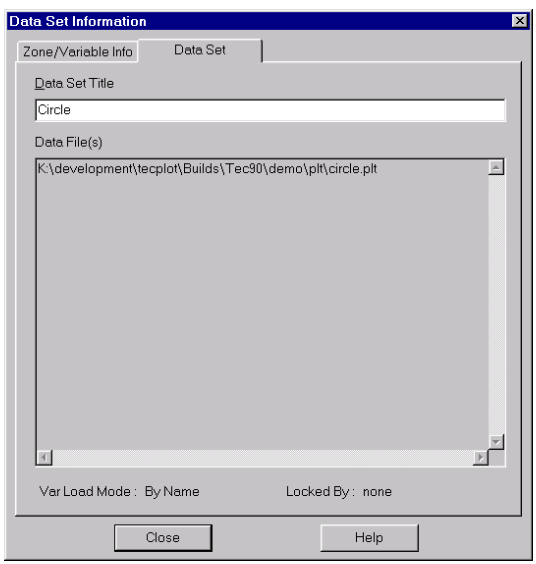




**Figure 4-20.** The Data Set Information dialog, showing the Zone/Variable Info (top) and Data Set page (bottom).

The following information is provided on two pages in the dialog. On the Zone/Variable Info page are:

- **Zone(s):** Lists all zones by number, with their titles. Select one zone to display its name in the Zone Name field, where the zone name can be modified.
- **Zone Name:** Enter a new name for a selected zone.
- **Zone Type (Ordered or FE data):** Displays the type of zone selected in the Zone(s) listing. For ordered data, it is followed by the index values for IMax, JMax and KMax (shown below). For finite-element data, it is followed by the element type, number of points, and number of elements (see below).
  - **IMax (Ordered data):** Displays the IMax value of the zone selected in the Zone(s) listing.
  - **JMax (Ordered data):** Displays the JMax value of the zone selected in the Zone(s) listing.
  - **KMax (Ordered data):** Displays the KMax value of the zone selected in the Zone(s) listing.
  - **Pts (FE data):** Displays the number of data points in the zone selected in the Zone(s) listing.
  - **Elem (FE data):** Displays the number of elements in the zone selected in the Zone(s) listing.
- **Variable(s):** Lists all variables by number, with their names. Select one variable to display its name in the Variable Name field, where the name can then be modified.
- **Variable Name:** Enter a new name for a selected variable.
- **Var Type:** Displays the type of data of the selected variable in the Variable(s) field.
- **Var Range -- Selected Zone:** Displays the Min and Max values for the selected variable in the selected zone.
- **Var Range -- Active Zone(s):** Displays the Min and Max values for the selected variable for all active zones.

On the Data Set page are:

- **Data Set Title:** Enter a title for the current data set, or edit an existing title. The default is the result of concatenating the titles specified in each **`Title`** record encountered in the data files making up the data set.
- **Data File(s):** Lists the names and paths of all external data files making up the current data set.
- **Var Load Mode:** Depending on the method used, this displays either By Position or By Name.

- **Locked By:** This field will inform you if the current data set has been locked by an add-on. Add-ons can lock a data set which in turn prevents your from deleting zones or deleting the last frame associated with the data set.

# CHAPTER 5 Formatting ASCII Data for Tecplot

This chapter tells you how to format data so your data files may be loaded directly into Tecplot. You can also load data generated by, or tabulated in, other software packages. Amtec has written some data loaders using Tecplot's Add-on Developer's Kit (ADK). These loaders convert data from a number of popular software packages into a format readable by Tecplot. They are described in Chapter 7, "Data Loaders: Tecplot's Import Feature." Tecplot users can also write loaders of their own using the ADK.

Data files read by Tecplot may be binary or ASCII. Reading an ASCII data file into Tecplot can be much slower than reading a binary data file, as binary data files take up less disk space. You can use Tecplot or Preplot to convert ASCII data files to binary. See Section 6.1, "Loading Tecplot-Format Data Files," for details on using Tecplot, or Section 5.5, "Converting ASCII Data Files to Binary," for details on using Preplot.

The following sections describe the format of ASCII data files. The documentation for the binary data file format is included as comments in the Preplot source code. If your data is generated by a computer program written in FORTRAN or C, you may be able to generate binary data files directly using the utilities described in Chapter 11, "Writing Binary Data for Loading into Tecplot," of the *Tecplot Reference Manual*.

## 5.1. ASCII Data File Records

An ASCII data file begins with an optional file header defining a title for the data file and or the names of the variables. The file header is followed by optional zone records which contain the plot data. Zone records may contain either ordered data or finite-element data. You may also include text records, geometry records, and custom-label records that create text, geometries, and/or custom labels on your plots. Each data file may have up to 32,700 zone records, up to ten custom label records, and any number of text records and geometry records. These records may be in any order.

The first line in a zone, text, geometry, or custom-label record begins with one of the keywords **ZONE**, **TEXT**, **GEOMETRY**, or **CUSTOMLABELS**. The maximum length of a line in a data file is 4,000 characters (unless you edit and recompile the Preplot source code). Any line may be continued onto one or more following lines (except for text enclosed in double quotes [**"**]). Double quotes must be used to enclose character strings with embedded blank spaces or other special characters. A backslash (****) may be used to remove the significance of (or escape) the next character (that is, **\"** produces a single double-quote). Any line that begins with an octothorp (**#**) is treated as a comment and ignored.

The following simple example of a Tecplot ASCII data file has one small zone and a single line of text:

```
TITLE="Simple Data File"
VARIABLES="X" "Y"
ZONE I=4 F=POINT
1 1
2 1
2 2
1 2
TEXT X=10 Y=90 T="Simple Text"
```

The format of the ASCII data file is summarized in Section 5.1.7, “Summary of Data File Records.”

### 5.1.1. File Header

In the file header of your data file, you may specify an optional title that is displayed in the headers of Tecplot frames. The title line begins with **TITLE=**, followed by the title text enclosed in double-quotes. You may also assign a name to each of the variables by including a line that begins with **VARIABLES=**, followed by each variable’s name enclosed in double quotes. The quoted variable names should be separated by spaces or commas. Tecplot calculates the number of variables (*N*) from the list of variable names. If you do not specify the variable names (and your first zone is in **POINT** or **FEPOINT** format), Tecplot sets the number of variables equal to the number of numeric values in the first line of zone data for the first zone, and names the variables **V1**, **V2**, **V3**, and so forth.

Initially, Tecplot uses the first two variables in data files as the X- and Y-coordinates, and the third variable for the Z-coordinate of 3-D plots. You may, however, order the variables in the data file any way you want, since you can interactively reassign the variables to the X-, Y-, and or Z-axes using Tecplot dialogs.

If the file header occurs in a place other than at the top of the data file, a warning is printed and the header is ignored. This allows you to concatenate two or more ASCII data files before using Tecplot (provided each data file has the same number of variables per data point).

### 5.1.2. Zone Records

A zone record consists of a control line that begins with the keyword "**`ZONE`**" followed by a set of numerical data called the zone data. The format of the zone control line is shown in Section 5.1.7, "Summary of Data File Records."

**5.1.2.1. The Format Parameter.** The zone data are in the format specified by the **`F`** (format) parameter in the control line. There are two basic types of zones: ordered and finite-element. Ordered zones have the formats **`POINT`** and **`BLOCK`**; finite-element zones have the formats **`FEPOINT`** and **`FEBLOCK`**. **`POINT`** format is assumed if the **`F`** parameter is omitted (thus, by default, zones are assumed to be ordered). See Section 5.2, "Ordered Data," for more information on ordered zones, and Section 5.3, "Finite-Element Data," for details on finite-element data.

In **`POINT`** and **`FEPOINT`** format, the values for all variables are given for the first point, then the second point, and so on. In **`BLOCK`** and **`FEBLOCK`** format, all of the values for the first variable are given in a block, then all of the values for the second variable, then all of the values for the third, and so forth. More detail on this is given below.

**5.1.2.2. A Simple Example of POINT Format.** If you have only one zone of data in **`POINT`** format, and it is one-dimensional (that is, *JMax=1*, *KMax =1*), you may omit the zone control line. If you want Tecplot to determine the number of variables, you may create a data file with only the zone data, such as the following:

```
12.5 23 45 1.
14.3 24 46 2.
12.2 24 50 3.
13.3 26 51 4.
13.5 27 55 5.
```

Tecplot calculates the number of data points (*IMax*) in the zone by assuming that each row represents a data point and each column represents a variable, and creates an I-ordered zone. This type of structure is good for XY-plots and scatter plots. If there are multiple zones, two- or three-dimensional zones, finite-element zones, or **`BLOCK`**-format zone data, you must include a zone control line at the beginning of each zone record.

**5.1.2.3. Data Types.** Each variable in each zone in the data file may have its own data type. Tecplot supports the following six data types:

- **`SINGLE`** (four-byte floating point values).
- **`DOUBLE`** (eight-byte floating point values).
- **`LONGINT`** (four-byte integer values).
- **`SHORTINT`** (two-byte integer values).

- **BYTE** (one-byte integer values, from 0 to 255).
- **BIT**.

The data type determines the amount of storage Tecplot assigns to each variable. Therefore, the lowest level data type should be used whenever possible. For example, imaging data, which usually consists of numerical values ranging from zero to 255, should be given a data type of **BYTE**. By default, Tecplot treats numeric data as data type **SINGLE**. If any variable in the zone uses the **BIT** data type, the zone format must be **BLOCK** or **FEBLOCK**; you cannot use **POINT** or **FEPOINT** format.

**5.1.2.4. Listing Your Data.** Numerical values in zone data must be separated by one or more spaces, commas, tabs, new lines, or carriage returns. Blank lines are ignored. Integer (**101325**), floating point (**101325.0**), and exponential (**1.01325E+05**) numbers are accepted. To repeat a particular number in the data, precede it with a repetition number as follows: "*Rep***Num*," where *Rep* is the repetition factor and *Num* is some numeric value to be repeated. For example, you may represent 37 values of 120.5 followed by 100 values of 0.0 as follows:

```
37*120.5, 100*0.0
```

**5.1.2.5. Zone Types and Their Control Lines.** As stated above, there are two distinct types of zones: ordered zones and finite-element zones. Ordered zones are I-, IJ-, and IJK-ordered zones (formats **POINT** and **BLOCK**). Finite-element zones are FE-surface and FE-volume zones (formats **FEPOINT** and **FEBLOCK**). The control lines for these zone types differ in the parameters needed. Both zone types can use the **C** (*color*), **F** (*format*), **T** (*zonetitle*), **D** (*duplist*), and **DT** (*datatype*) parameters, although the format of the **F** and **D** parameters is slightly different for each zone type.

The **T** parameter specifies a title for the zone. This may be any text string up to 64 characters in length. If you supply a longer text string, it is automatically truncated to the first 64 characters. The titles of zones appear in the Plot Attributes and other dialogs, and, optionally, in the XY-plot legend. (You can use keywords in the zone titles to identify sets of zones to enable/disable or to change zone attributes.) The **C** parameter sets an initial color for the zone. This may be overridden interactively, or by use of a stylesheet. The **DT** (*type1*, *type2*, *type3*, ...) parameter specifies the data types for the variables in a zone.

The **D** (*duplist*) parameter specifies a list of variables to duplicate from the preceding zone, which must have the same dimensions (*IMax*, *JMax*, and *KMax*) as the new zone. If a variable is duplicated using the **D** parameter in the zone control line, no values are listed for this variable, and the values of the specified variables are obtained from the previous zone record. For example, if the zone control line has **D=(1,2,4)**, the first values listed would be for variable **V3**, the second, for variable **V5**.

For ordered zones, you may specify the **I** (*IMax*), **J** (*JMax*), and **K** (*KMax*) parameters, which store the number of data points in the I, J, and K directions. **J** and **K** both default to 1. **I** must be specified if **J** is used; **I** and **J** must be specified if **K** is used. If all are omitted, Tecplot assumes an I-ordered zone and calculates *IMax* for you.

**Note:** **I** and **J** are not equivalent to either the number of variables or the number of data points. The number of data points is equal to the product of **I**, **J**, and **K**.

For finite-element zones, described in Section 5.3, “Finite-Element Data,” you must specify the **N** (*numnodes*) and may optionally include the **ET** (*elementtype*), the **E** (*numelements*), and or the **NV** (*nodevalue*) parameter. If the **E** parameter is not specified, Tecplot calculates it from the number of node sets in the connectivity list following the node data. The **NV** (*nodevalue*) parameter specifies the number of variables which represent the “Node” value in FE data.

The **D** (*duplist*) parameter specifies a list of variables to duplicate and/or the keyword **FECONNECT**, which duplicates the connectivity list of the preceding zone. The preceding zone must have the same *numnodes* and *numelements* as the new zone in order to use the **D** parameter.

The following sections give simple examples of zone data in various formats, as well as sample pieces of FORTRAN code that you can use as templates to print out your own data. Note that the sample code is intended only as a general example—the zone data that it produces contains only one value per line. You may want to modify the code to suit your own needs.

### 5.1.3. Text Record

Text records are used to import text directly from a data file. Text can also be imported into Tecplot using a macro file. Text may be titles, labels, or other information. You may create data files containing only text records and read them into Tecplot just as you would read any other data file. You may delete and edit text originating from data files just like the text that you create interactively.

The text record consists of a single control line. The control line starts with the keyword **TEXT** and has one or more options:

- The text string is defined in the required **T** (*text*) parameter.
- The color is controlled by the **C** (*color*) parameter.
- Use the **CS** (*coordinatesys*) parameter to specify the text coordinate system, either **FRAME** or **GRID**. If you specify the frame coordinate system (the default), the values of the **X** (*xorigin*) and **Y** (*yorigin*) parameters are in frame units; if you specify grid coordinates, **X** and **Y** are in grid units (i.e., units of the physical coordinate system). **X** and **Y** locate the anchor point of the text string.
- Use the **AN** (*textanchor*) parameter to specify the position of the anchor point relative to the text. There are nine possible anchor positions, as shown in Figure 5-1.

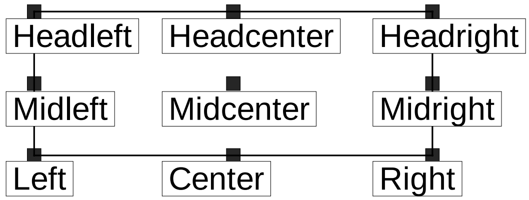



**Figure 5-1.** Text anchor positions—values for the **AN** parameter.

- Use the **HU** (*heightunits*) parameter to assign units for character heights. If the **CS** parameter is **FRAME**, you can set **HU** to either **FRAME** or **POINT**. If the **CS** parameter is **GRID**, you can set **HU** to either **GRID** or **FRAME**.
- Use the **H** parameter to specify the height; it is measured in the units defined by the **HU** parameter.
- To include multiple lines of text in a single text record, include **\\n** in the text string to indicate a new line.
- You can assign the line spacing for multi-line text using the **LS** (*linespacing*) parameter. The default value, 1, gives single-spacing. Use 1.5 for line-and-a-half spacing, 2 for double-spacing, and so on.

You may optionally draw a box around the text string using the **BX** (*boxtype*) parameter. The parameters **BXO** (*boxoutlinecolor*), **BXM** (*boxmargin*), and **LT** (*linethickness*) are used if the *boxtype* is **HOLLOW** or **FILLED**. The parameter **BXF** (*boxfillcolor*) is used only if the *boxtype* is **FILLED**. The default *boxtype*, **NOBOX**, ignores all other *box* parameters.

The **S** (*scope*) parameter specifies the text scope. **GLOBAL** scope is the same as selecting the check box Show in “Like” Frames in the Text Options dialog. See Section 18.1.6.3, “Specifying the Scope of the Text,” for details.

You may also use the **ZN** (*zone*) parameter to attach text to a specific zone or XY mapping. For further information, see Section 16.1.6.4, "Attaching Text to Zones or X-Y Mapping."

**5.1.3.1. Examples of Text Records.** You may attach a macro command to the text with the **MFC** parameter. See Section 18.5, “Linking Text and Geometries to Macros.”

Some simple examples of text records are shown below. The first text record specifies only the origin and the text. The next text record specifies the origin, color, font, and the text. The last text record specifies the origin, height, box attributes, and text. Note that the control line for the text can span multiple file lines if necessary (as in the last text record below).

```
TEXT X=50, Y=50, T="Example Text"

TEXT X=10, Y=10, F=TIMES-BOLD, C=BLUE, T="Blue Text"

TEXT X=25, Y=90, CS=FRAME, HU=POINT, H=14,
    BX=FILLED, BXF=YELLOW, BXO=BLACK, LS=1.5,
    T="Box Text \\n Multi-lined text"
```

### 5.1.4. Geometry Record

Geometry records are used to import geometries from a data file. Geometries are line drawings that may be boundaries, arrows, or even representations of physical structures. You may create data files containing only geometry and text records and read them into Tecplot. You may delete and edit geometries originating from data files just like the geometries that you create interactively.

The geometry record control line begins with the keyword **GEOMETRY**. Use the **CS** (*coordinatesys*) parameter to specify the geometry coordinate system, either **FRAME** or **GRID**. If you specify the frame coordinate system (the default), the values of the **X** (*xorigin*) and **Y** (*yorigin*) parameters are in frame units; if you specify grid coordinates, **X** and **Y** are in grid units (that is, units of the physical coordinate system). **X** and **Y** locate the anchor point, or origin, of the geometry, which is the center of a circle or ellipse, the lower left corner of a square or rectangle, and the anchor point of a polyline. The anchor point specifies the offset of all the points: if *X=1*, *Y=1*, and the first point is (1, 2), and the second point is (2, 4), then Tecplot draws at (2, 3) (1+1, 2+1) then (3, 5) (2+1, 4+1). In other words, the points for any geometry are always relative to the specified anchor point. The **Z** (*zorigin*) is specified only for **LINE3D** geometries, and, since **LINE3D** geometries are always in grid mode, **Z** is always in units of the Z-axis.

Geometry types are selected with the **T** (*geomtype*) parameter. The available geometry types are listed below:

- **SQUARE**: A square with lower left corner at **X**, **Y**.
- **RECTANGLE**: A rectangle with lower left corner at **X**, **Y**.
- **CIRCLE**: A circle centered at **X**, **Y**.
- **ELLIPSE**: An ellipse centered at **X**, **Y**.
- **LINE**: A set of 2-D polylines (referred to as multi-polylines) anchored at **X**, **Y**.
- **LINE3D**: A set of 3-D polylines (referred to as multi-polylines) anchored at **X**, **Y**, **Z**.

The color of the geometry is controlled by the **C** (*color*) parameter. Any geometry type except **LINE3D** may be filled with a color by using the **FC** (*fillcolor*) parameter. With both **C** (*color*) and **FC** (*fillcolor*) on the control line, the geometry is outlined in one color and filled with another. Each polyline of a **LINE** geometry is filled individually (by connecting the last point

of the polyline with the first). Not specifying the **FC** (*fillcolor*) parameter results in a hollow, or outlined, geometry drawn in the color of the **C** (*color*) parameter.

You can control how geometries are drawn using the **L** (*linetype*), **LT** (*linethickness*), and **PL** (*patternlength*) parameters. You can set **L** to any of Tecplot's line patterns (**SOLID**, **DASHED**, **DOTTED**, **DASHDOT**, **LONGDASH**, **DASHDOTDOT**). You can set **LT** and **PL** to any value, using frame units.

The control line of the geometry is followed by geometry data. For **SQUARE**, the geometry data consists of just one number: the side length of the square.

For **RECTANGLE**, the geometry data consists of two numbers: the first is the width (horizontal axis dimension), and the second is the height (vertical axis dimension).

For **CIRCLE**, the geometry data is one number: the radius. For **ELLIPSE**, the geometry data consists of two numbers: the first is the horizontal axis length and the second is the vertical axis length. For both circles and ellipses, you can use the **EP** (*numellipsepts*) parameter to specify the number of points used to draw circles and ellipses. All computer-generated curves are simply collections of very short line segments; the **EP** parameter allows you to control how many line segments Tecplot uses to approximate circles and ellipses. The default is 72.

For **LINE** and **LINE3D** geometries, the geometry data is controlled by the **F** (*format*) parameter. These geometries may be specified in either **POINT** or **BLOCK** format. By default, **POINT** format is assumed. Each geometry is specified by the total number of polylines, up to a maximum of 50. Each polyline is defined by a number of points and a series of XY- or XYZ-coordinate points between which the line segments are drawn. In **POINT** format, the XY- or XYZ-coordinates are given together for each point. In **BLOCK** format, all the X-values are listed, then all the Y-values, and (for **LINE3D** geometries) all the Z-values. All coordinates are relative to the **X**, **Y**, and **Z** specified on the control line. You can specify points in either single or double precision by setting the **DT** (*datatype*) parameter to either **SINGLE** or **DOUBLE**.

For **LINE** geometries, you can specify arrowheads using the **AAT** (*arrowheadattach*), **AST** (*arrowheadstyle*), **ASZ** (*arrowheadsize*), and **AAN** (*arrowheadangle*) parameters. See Section 5.1.7, "Summary of Data File Records," for details. These parameters provide the same functionality available when you create a line geometry interactively.

The **S** (*scope*) parameter specifies the geometry's scope. **GLOBAL** scope is the same as selecting the check box Show in Like Frames in the Geometry dialog. See Section 18.2.2.6, "Specifying Geometry Scope," for details.

You may also use the **ZN** (*zone*) parameter to attach geometry to a specific zone or XY-mapping.

You may attach a macro command to the text with the **MFC** parameter. See Section 18.5, "Linking Text and Geometries to Macros."

**LINE3D** geometries must be created in a data file. They may not be created interactively. **LINE3D** geometries are always in grid mode. To view **LINE3D** geometries in Tecplot, you must be in 3D frame mode, which requires at least one zone. Thus, a data file with only **LINE3D** geometries is useful only as a supplement to other data files.

**5.1.4.1. Examples of Geometry Records.** The following geometry record defines a rectangle of **40** width and **30** height:

```
GEOMETRY  T=RECTANGLE
40 30
```

The following geometry record defines an origin and a red circle of **20** radius, with an origin of (**75**, **75**) that is filled with blue:

```
GEOMETRY X=75, Y=75, T=CIRCLE, C=RED, FC=BLUE,CS=FRAME
 20
```

The following geometry record defines an origin and two polylines, drawn using the Custom 3 color. The first polyline is composed of three points, the second of two points.

```
GEOMETRY X=50, Y=50, T=LINE, C=CUST3
 2
 3
 0 1
 0 0
 2 0
 2
 0 0
 1 2
```

In **BLOCK** format, the same geometry looks like the following:

```
GEOMETRY X=50, Y=50, T=LINE, C=CUST3, F=BLOCK, CS=FRAME
 2
 3
 0 0 2
 1 0 0
 2
 0 1
 0 2
```

The next geometry record defines a purple ellipse with a horizontal axis length of **20** and a vertical axis length of **10**, with an origin of (**10, 70**), that is filled with yellow.

```
GEOMETRY X=10, Y=70, T=ELLIPSE, C=PURPLE, FC=YELLOW
 20 10
```

The final geometry record is a 3-D polyline with four points that is composed of one polyline using the default origin of (**0, 0, 0**):

```
GEOMETRY T=LINE3D
 1
 4
 0 0 0
 1 2 2
 3 2 3
 4 1 2
```

In **BLOCK** format, this geometry record can be written as follows:

```
GEOMETRY T=LINE3D, F=BLOCK
 1
 4
 0 1 3 4
 0 2 2 1
 0 2 3 2
```

### 5.1.5. A More Extensive Example of a Geometry Record

In the **TextGeom** file shown below, there are four text records (showing a circle, ellipse, rectangle, and line). A plot of the file is shown in Figure 5-2.

```
TEXT X=20, Y=85, F=HELV-BOLD, C=BLUE, H=7.5,
      T="Example Text"
TEXT X=20, Y=75, F=TIMES-BOLD, H=5, T="Subtitle"
TEXT X=80, Y=25, F=TIMES-ITALIC-BOLD, H=4, C=RED,
      BX=FILLED, BXF=YELLOW, BXM=50, BXO=CYAN,
      T="Filled Box"
TEXT X=41, Y=8, H=4, F=COURIER-BOLD,
      C=CUST3, BX=HOLLOW, BXO=CUST4, T="Hollow Box"
GEOMETRY X=50, Y=50, T=RECTANGLE, FC=WHITE, C=BLUE
   40 30
GEOMETRY X=30, Y=30, T=CIRCLE, FC=BLUE, C=GREEN
   20
GEOMETRY X=70, Y=65, T=LINE, FC=PURPLE, C=BLACK
   1
   4
   -10 0
   0 10
   010 10
   10 0.6
GEOMETRY T=LINE, C=CUST1
   2
   3
   5 50
```

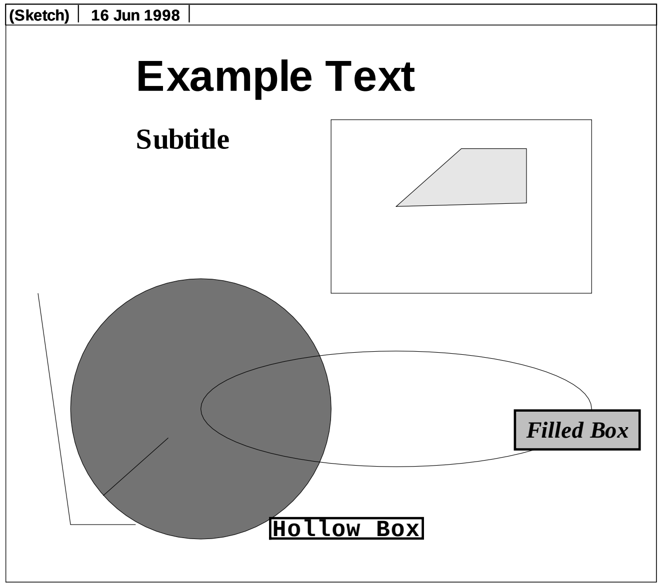



**Figure 5-2.** Text and geometries created from the sample in Section 5.1.5, “A More Extensive Example of a Geometry Record.”

```
   10 10
   20 10
   2
   15 15
   25 25
GEOMETRY X=60, Y=30, T=ELLIPSE, C=CUST8
   30 10
```

### 5.1.6. Custom Label Record

The custom label record is an optional record to define sets of text strings for use in custom labeling the values of an axis, contour legend or value labels, or variable-value node labels. The custom-label record begins with the keyword **CUSTOMLABELS**, followed by one or more text strings. The text strings must be enclosed within double quotes (“**"**”) if they contain any commas, spaces or other special characters, or if they might be confused with valid data file keywords. Enclosing the strings in double quotes is always recommended.

The first custom-label string corresponds to a value of one on the axis, the next to a value of two, the next to a value of three, and so forth. Custom labels may appear one to a line, or there

may be more than one on a line, separated by a comma or space. Multiple custom-label records can be present in a data file. If this is the case, you choose which set to assign to a given axis, contour legend, or variable-value node labels. Custom labels are discussed in more detail in Section 17.5.2, “Controlling Tick Mark Labels.”

A simple example of a custom-label record is shown below. **`MON`** corresponds to a value of **`1`**, **`TUE`** corresponds to **`2`**, **`WED`** to **`3`**, **`THU`** to **`4`**, and **`FRI`** to **`5`**. Since custom labels have a wrap-around effect, **`MON`** also corresponds to the values **`6`**, **`11`**, and so forth.

```
CUSTOMLABELS "MON", "TUE", "WED", "THU", "FRI"
```

### 5.1.7. Summary of Data File Records

The following table summarizes the records and parameters allowable in Tecplot data files.

| Data File Section | Records | Parameter Descriptions |
|---|---|---|
| File Header | **`TITLE`** = "*filetitle*"<br>**`VARIABLES`** =<br>"*vname1*"<br>"*vname2*" ... | Title for data file.<br><br>Name of first variable.<br>Name of second variable. |
| Ordered Zone Record | **`ZONE`**<br>**`T`**="*zonetitle*"<br>**`I`**=*IMax*<br>**`J`**=*JMax*<br>**`K`**=*KMax*<br>**`C`**=*color*<br><br><br>**`F`**=*orderedformat*<br>**`D`**=(*duplist*)<br><br>**`DT`**=(*datatypelist*) | <br>Title for zone.<br>Number of points in I-direction.<br>Number of points in J-direction.<br>Number of points in K-direction.<br>One of the following: **`BLACK`**, **`RED`**, **`GREEN`**, **`BLUE`**, **`CYAN`**, **`YELLOW`**, **`PURPLE`**, **`WHITE`**, **`CUST1`**, **`...`**, **`CUST8`**.<br>Either **`POINT`** or **`BLOCK`**.<br>List of variables to duplicate from previous zone.<br>List specifying data type for each variable, from among the following: **`SINGLE`**, **`DOUBLE`**, **`LONGINT`**, **`SHORTINT`**, **`BYTE`**, **`BIT`**. |

| Data File Section | Records | Parameter Descriptions |
| --- | --- | --- |
| Finite-element Zone Record | **ZONE** | |
| | **T="***zonetitle***"** | Title for zone. |
| | **N=***numnodes* | Number of nodes. |
| | **E=***numelements* | Number of elements. |
| | **ET=***elementtype* | One of the following: **TRIANGLE**, **QUADRILATERAL**, **TETRAHEDRON**, **BRICK**.. |
| | **C=***color* | See description above. |
| | **F=***feformat* | Either **FEPOINT** or **FEBLOCK**. |
| | **D=(***feduplist***)** | List of variables to duplicate from previous zone, and/or the keyword **FECONNECT**. |
| | **NV=***nodevariable* | Which variable represents the Node value. |
| | **DT=(***datatypelist***)** | See description above. |

| Data File Section | Records | Parameter Descriptions |
|---|---|---|
| Text Record | **TEXT** | |
| | **X**=*xorigin* | *X* origin of object in *coordinatesys* units. |
| | **Y**=*yorigin* | *Y* origin of object in *coordinatesys* units. |
| | **F**=*font* | One of the following: **HELV**, **HELV-BOLD**, **TIMES**, **TIMES-ITALIC**, **TIMES-BOLD**, **TIMES-ITALIC-BOLD**, **COURIER**, **COURIER-BOLD**, **GREEK**, **MATH**, **USER-DEF**. |
| | **CS**=*coordinatesys* | Either **FRAME** or **GRID**. |
| | **HU**=*heightunits* | In **FRAME** *coordinatesys*, either **FRAME** or **POINT**; in **GRID** *coordinatesys*, either **GRID** or **FRAME**. |
| | **AN**=*textanchor* | One of the following: **LEFT**, **CENTER**, **RIGHT**, **MIDLEFT**, **MIDCENTER**, **MIDRIGHT**, **HEADLEFT**, **HEADCENTER**, **HEADRIGHT**. |
| | **C**=*color* | See description above. |
| | **A**=*angle* | Angle in degrees, counter-clockwise from horizontal. |
| | **H**=*height* | Character height in *heightunits*. |
| | **LS**=*linespacing* | Line spacing for multiple-line text. |
| | **S**=*scope* | Either **LOCAL** or **GLOBAL**. |
| | **T="***text***"** | Alphanumeric text string. |
| | **BX**=*boxtype* | One of **NOBOX**, **HOLLOW**, or **FILLED**. |
| | **BXM**=*boxmargin* | Margin around text as fraction of text height. |
| | **BXF**=*boxfillcolor* | Fill color for box; use *color* options. |
| | **BXO**=*boxcolor* | Color of text box outline; use *color* options. |
| | **LT**=*boxlinethickness* | Line thickness of text box. |
| | **ZN**=*zone* | Zone (or XY-mapping) number to which this item is assigned. |
| | **MFC="***macrofunctioncommand***"** | Macro function command. |

| Data File Section | Records | Parameter Descriptions |
|---|---|---|
| Geometry Record | **GEOMETRY** | |
| | **X**=*xorigin* | *X*-origin of object in *coordinatesys* units. |
| | **Y**=*yorigin* | *Y*-origin of object in *coordinatesys* units. |
| | **Z**=*zorigin* | *Z*-origin of object in *coordinatesys* units. |
| | **CS**=*coordinatesys* | Either **FRAME** or **GRID**. |
| | **C**=*color* | See description above. |
| | **L**=*linetype* | One of the following: **SOLID**, **DASHED**, **DASHDOT**, **DOTTED**, **LONGDASH**, **DASHDOTDOT**. |
| | **PL**=*patternlength* | Pattern length for specified line type. |
| | **LT**=*linethickness* | Line thickness for geometry outline. |
| | **T**=*geomtype* | One of the following: **LINE**, **SQUARE**, **RECTANGLE**, **CIRCLE**, **ELLIPSE**, **LINE3D**. |
| | **EP**=*numellipsepts* | Number of points to use to approximate circles and ellipses. |
| | **AST**=*arrowheadstyle* | One of **PLAIN**, **HOLLOW**, or **FILLED**. |
| | **AAT**=*arrowheadattach* | One of the following: **NONE**, **BEGINNING**, **END**, **BOTH**. |
| | **ASZ**=*arrowheadsize* | Size of arrowhead in Frame units. |
| | **AAN**=*arrowheadangle* | Angle of arrowhead in degrees. |
| | **DT**=*datatype* | Either **SINGLE** or **DOUBLE** (applies to 2- and 3-D polylines only). |
| | **S**=*scope* | Either **LOCAL** or **GLOBAL**. |
| | **F**=*geomformat* | Either **POINT** or **BLOCK**. |
| | **FC**=*geomfillcolor* | Fill color for geometry; use *color* options |
| | **ZN**=*zone* | Zone (or XY-mapping) number to which this geometry is assigned. |
| | **MFC="***macrofunctioncommand***"** | Macro function command. |
| Custom Labels Record | **CUSTOMLABELS** | |
| | **"***label1***"** | String for value of one when using custom labels. |
| | **"***label2***" ...** | String for value of two when using custom labels. |

## 5.2. Ordered Data

For ordered data, the numerical values in the zone data must be in either **POINT** or **BLOCK** format, determined by the **F** (*format*) parameter.

### 5.2.1. I-Ordered Data

I-ordered data has only one index, the I-index. This type of data is typically used for XY-plots, scatter plots, and irregular (random) data for triangulation or for interpolation into an IJ-or IJK-ordered zone within Tecplot.

In I-ordered data, the I-index varies from one to *IMax*. The total number of data points is *IMax*. The total number of values in the zone data is *IMax*N* (where *N* is the number of variables). For data in **POINT** format, *IMax* is calculated by Tecplot from the zone data if it is not explicitly set by the zone control line (using the **I**-parameter).

**5.2.1.1. Example of I-Ordered Data in POINT Format.** A simple example of I-ordered data in **POINT** format is listed below. There are two variables (**X**, **Y**) and five data points. In this example, each row of data corresponds to a data point and each column to a variable. This data set is plotted in Figure 5-3; each data point is labeled with its I-index.

```
VARIABLES = "X","Y"
ZONE I=5, F=POINT
2    4
3    9
5   25
6   36
7   49
```

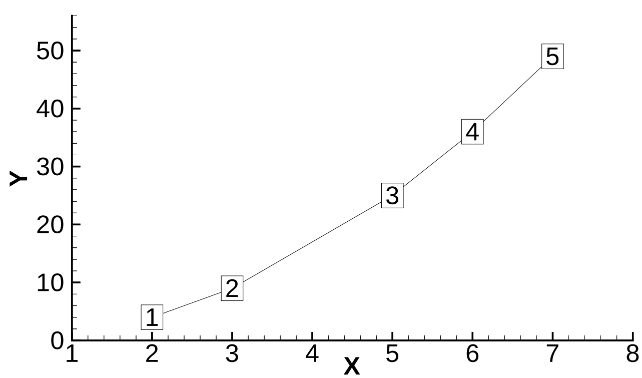



**Figure 5-3.** An I-ordered data set.

**5.2.1.2. FORTRAN Code Example to Generate I-Ordered Data in POINT Format.** The following sample FORTRAN code shows how to create I-ordered data in **POINT** format:

```
  INTEGER VAR
    .
    .
    .
  WRITE (*,*) ´ZONE F=POINT, I=´, IMAX
  DO 1 I=1,IMAX
      DO 1 VAR=1,NUMVAR
1         WRITE (*,*) ARRAY(VAR,I)
```

**5.2.1.3. Example of I-Ordered Data in BLOCK Format.** The same data as in Section 5.2.1.1. is shown below in **BLOCK** format. In this example, each column of zone data corresponds to a data point; each row to a variable.

```
VARIABLES = "X", "Y"
ZONE I=5, F=BLOCK
2 3 5 6 7
4 9 25 36 49
```

In **BLOCK** format all *IMax* values of each variable are listed, one variable at a time.

**5.2.1.4. Example of FORTRAN Code to Generate I-Ordered Data in BLOCK Format.** The following sample FORTRAN code shows how to create I-ordered data in **BLOCK** format:

```
  INTEGER VAR
    .
    .
    .
  WRITE (*,*) ´ZONE F=BLOCK, I=´, IMAX
  DO 1 VAR=1,NUMVAR
      DO 1 I=1,IMAX
1         WRITE (*,*) ARRAY(VAR,I)
```

**5.2.1.5. Example: A Multi-zone XY-Plot.** The two tables below show the values of pressure and temperature measured at four locations on some object at two different times. The four locations are different for each time measurement.

| Time = 0.0 seconds: | | |
|---|---|---|
| **Position** | **Temperature** | **Pressure** |
| 71.30 | 563.7 | 101362.5 |
| 86.70 | 556.7 | 101349.6 |
| 103.1 | 540.8 | 101345.4 |
| 124.4 | 449.2 | 101345.2 |

| Time = 0.1 seconds: | | |
|---|---|---|
| **Position** | **Temperature** | **Pressure** |
| 71.31 | 564.9 | 101362.1 |
| 84.42 | 553.1 | 101348.9 |
| 103.1 | 540.5 | 101344.0 |
| 124.8 | 458.5 | 101342.2 |

For this case, we want to set up two zones in the data file, one for each time value. Each zone has three variables (**`Position`**, **`Temperature`**, and **`Pressure`**) and four data points (one for each location). This means that *IMax=4* for each zone. We include a text record (discussed in Section 5.1.3., “Text Record”) to add a title to the plot. A data file in **`POINT`** format is given below. The plot shown in Figure 5-4 can be produced from this file.

```
TITLE = "Example: Multi-zone XY-Plot"
VARIABLES = "Position", "Temperature", "Pressure"
ZONE T="0.0 seconds", I=4
71.30 563.7 101362.5
86.70 556.7 101349.6
103.1 540.8 101345.4
124.4 449.2 101345.2
ZONE T="0.1 seconds", I=4
71.31 564.9 101362.1
84.42 553.1 101348.9
103.1 540.5 101344.0
124.8 458.5 101342.2
TEXT CS=GRID, HU=GRID, X=0.36, Y=0.87, H=0.04, T="SAMPLE CASE"
```

A data file in **`BLOCK`** format is shown below. All of the values for the first variable (**`Position`**) at each data point are listed first, then all of the values for the second variable (**`Temperature`**) at each data point, and so forth.

```
TITLE = "Example: Multi-zone XY-Plot"
VARIABLES = "Position", "Temperature", "Pressure"
ZONE F=BLOCK, T="0.0 seconds", I=4
71.30 86.70 103.1 124.4
563.7 556.7 540.8 449.2
101362.5 101349.6 101345.4 101345.2
```

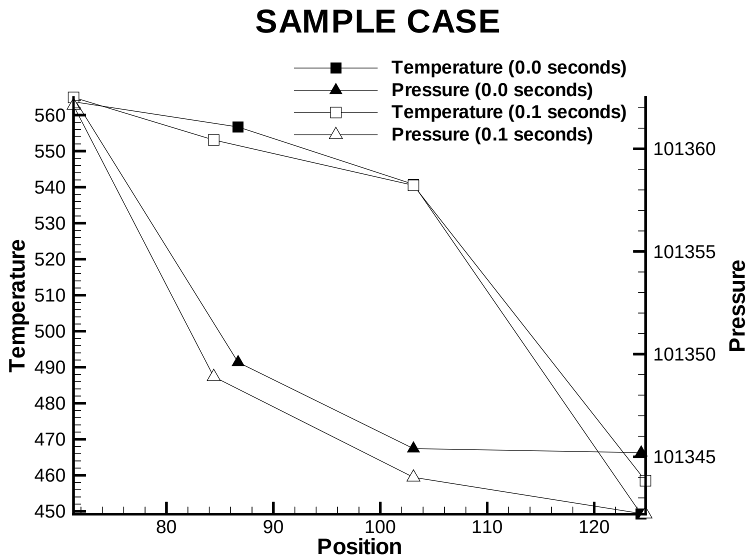



**Figure 5-4.** A multi-zone XY-plot.

```
ZONE F=BLOCK, T="0.1 seconds", I=4
71.31 84.42 103.1 124.8
564.9 553.1 540.5 458.5
101362.1 101348.9 101344.0 101342.2
TEXT CS=GRID, HU=GRID, X=0.36, Y=0.87, H=0.04, T="SAMPLE CASE"
```

A more compact data file for this example is in the point format shown below. Tecplot determines the number of variables from the number of values in the first line of data under the first zone. The variables and zones are assigned default names.

```
ZONE
71.30 563.7 101362.5
86.70 556.7 101349.6 103.1 540.8 101345.4 124.4 449.2 101345.2
ZONE
71.31 564.9 101362.1 84.42 553.1 101348.9 103.1 540.5 101344.0
124.8 458.5 101342.2
TEXT CS=GRID, HU=GRID, X=0.36, Y=0.87, H=0.04, T="SAMPLE CASE"
```

### 5.2.2. IJ-Ordered Data

IJ-ordered data has two indices: I and J. IJ-ordered data is typically used for 2- and 3-D surface mesh, contour, vector, and shade plots, but it can also be used to plot families of lines in XY-plots. See Chapter 8, “XY-Plots,” for more information. In IJ-ordered data, the I-index varies from 1 to *IMax*, and the J-index varies from one to *JMax*. The total number of data points is *IMax*JMax*. The total number of numerical values in the zone data is *IMax*JMax*N* (where *N* is the number of variables). Both *IMax* and *JMax* must be specified in the zone control line (with the **`I`** and **`J`** parameters). The I- and J-indices should not be confused with the X- and Y-coordinates—on occasions the two may coincide, but this is not the typical case.

The I-index varies the fastest. That is, when you write programs to print IJ-ordered data, the I-index is the inner loop and the J-index is the outer loop. Note the similarity between I-ordered data and IJ-ordered data with *JMax=1*.

**5.2.2.1. Example of IJ-Ordered Data in POINT Format.** A simple example of IJ-ordered data in **`POINT`** format is listed below. There are four variables (**`X`**, **`Y`**, **`Temperature`**, **`Pressure`**) and six data points. In this example, each row of data corresponds to a data point; each column to a variable. The first two lines are for *J=1*, the next two for *J=2*, the last two for *J=3*. The first, third, and fifth lines are for *I=1*; the second, fourth, and sixth lines are for *I=2*. This data is plotted in Figure 5-5; each data point is labeled with its IJ-index.

```
VARIABLES = "X", "Y", "Temperature", "Pressure"
ZONE I=2, J=3, F=POINT
3 0 0 50
7 2 0 43
2 4 1 42
6 6 0 37
1 8 1 30
5 9 1 21
```

**5.2.2.2. Example of FORTRAN Code to Generate IJ-Ordered Data in POINT Format.** The following sample FORTRAN code shows how to create IJ-ordered data in **`POINT`** format:

```
WRITE (*,*) ´VARIABLES = "X", "Y", "Temperature", "Pressure"’
WRITE (*,*) ´ZONE I=’, IMAX, ’, J=’, JMAX, ’, ’F=POINT’
 DO 1 J=1,JMAX
    DO 1 I=1, IMAX
1            WRITE (*,*) X(I,J), Y(I,J), T(I,J), P(I,J)
```

**5.2.2.3. Example of IJ-Ordered Data Set in BLOCK Format.** The same data set as in Section 5.2.2.1. is shown in **`BLOCK`** format below. In this example, each column of data corresponds to a data point; each row to a variable.

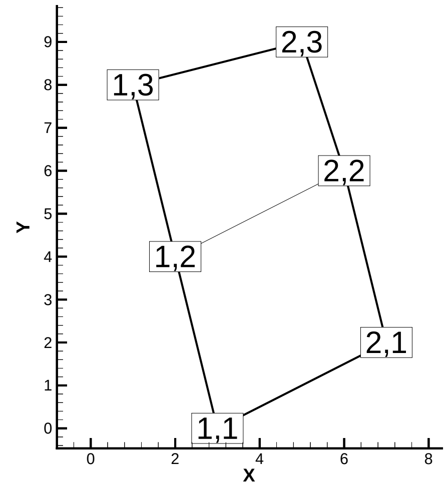



**Figure 5-5.** An IJ-ordered data set.

```
VARIABLES = "X", "Y", "Temperature", "Pressure"
ZONE I=2, J=3, F=BLOCK
3 7 2 6 1 5
0 2 4 6 8 9
0 0 1 0 1 1
50 43 42 37 30 21
```

In **BLOCK** format, all *IMax*JMax* values of each variable are listed, one variable at a time. Within each variable block, all the values of a variable at each data point are listed.

**5.2.2.4. Example FORTRAN Code to Generate IJ-Ordered Data in BLOCK Format.** The following sample FORTRAN code shows how to create IJ-ordered data in **BLOCK** format:

```
 INTEGER VAR
   .
   .
   .
 WRITE (*,*) ´ZONE F=BLOCK, I=´, IMAX,  ´, J=´, JMAX
 DO 1 VAR=1,NUMVAR
     DO 1 J=1,JMAX
         DO 1 I=1,IMAX
1             WRITE (*,*) ARRAY(VAR,I,J)
```

### 5.2.3. IJK-Ordered Data

IJK-ordered data has three indices: I, J, and K. This type of data is typically used for 3-D volume plots, although planes of the data can be used for 2- and 3-D surface plots. See Chapter 21, “Working with 3-D Volume Data,” for more information.

In IJK-ordered data, the I-index varies from 1 to *IMax*, the J-index varies from one to *JMax*, and the K-index varies from one to *KMax*. The total number of data points is *IMax*JMax*KMax*. The total number of values in the zone data is *IMax*JMax*KMax*N*, where *N* is the number of variables. The three indices, *IMax*, *JMax*, and *KMax*, must be specified in the zone control line using the **`I`**-, **`J`**-, and **`K`**-parameters.

The I-index varies the fastest; the J-index the next fastest; the K-index the slowest. That is, if you write a program to print IJK-ordered data, the I-index is the inner loop, the K-index is the outer loop, and the J-index is the loop in between. Note the similarity between IJ-ordered data and IJK-ordered data with *KMax=1*.

#### 5.2.3.1. An Example of IJK-Ordered Data in POINT Format.

A simple example of IJK-ordered data in **`POINT`** format is listed below. There are four variables (**`X`**, **`Y`**, **`Z`**, **`Temperature`**) and twelve data points. For this example, each row of data corresponds to a data point; each column to a variable. This data is plotted in Figure 5-6; each data point is labeled with its IJK-index.

```
VARIABLES = "X" "Y" "Z" "Temp"
ZONE I=3, J=2, K=2,F=POINT

 0 0 0 0
 3 0 1 5
 6 0 3 10
 0 6 3 10
 3 6 4 41
 6 6 6 72
 0 0 8 0
 3 0 9 29
 6 0 11 66
 0 6 11 66
 3 6 12 130
 6 6 14 169
```

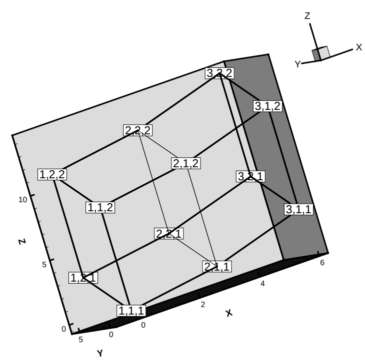



**Figure 5-6.** An IJK-ordered data set.

**5.2.3.2. The Same Data in BLOCK Format.** The same data set as Section 5.2.3.1., this time in `BLOCK` format, is shown below. For this example, each column of data corresponds to a data point; each row to a variable.

```
VARIABLES = "X" "Y" "Z" "Temp"
ZONE I=3, J=2, K=2, F=BLOCK

 0 3 6 0 3 6 0 3 6 0 3 6
 0 0 0 6 6 6 0 0 0 6 6 6
 0 1 3 3 4 6 8 9 11 11 12 14
 0 5 10 10 41 72 0 29 66 66 130 169
```

**5.2.3.3. An Example of FORTRAN Code to Generate an IJK-Ordered Zone in POINT Format.** The following sample FORTRAN code shows how to create an IJK-ordered zone in `POINT` format:

```
 WRITE (*,*) ’VARIABLES = "X", "Y", "Z", "Temp"’

 WRITE (*,*) ’ZONE I=’,IMAX,’ J=’,JMAX,’ K=’,KMAX,’ F=POINT’

DO 1 K=1,KMAX

   DO 1 J=1,JMAX

      DO 1 I=1,IMAX

1         WRITE (*,*) X(I,J,K), Y(I,J,K), Z(I,J,K), Temp(I,J,K)
```

In `BLOCK` format, all *IMax*JMax*KMax* values of each variable are listed, one variable at a time. Within each variable block, all the values of the variable at each data point are listed.

**5.2.3.4. An Example of FORTRAN Code to Generate IJK-Ordered Data in BLOCK Format.** The following sample FORTRAN code shows how to create an IJK-ordered zone in `BLOCK` format:

```
 INTEGER VAR
   .
   .
   .
   .
 WRITE (*,*) ´ZONE F=BLOCK, I=´, IMAX, ´, J=´, JMAX, ´, K=´, KMAX
 DO 1 VAR=1,NUMVAR
      DO 1 K=1,KMAX
         DO 1 J=1,JMAX
             DO 1 I=1,IMAX
1                WRITE (*,*) ARRAY(VAR,I,J,K)
```

### 5.2.4. One Variable Data Files

For ordered data, it is possible to read in a data file that has only one variable. Tecplot then creates the other required variables. That is, if your data is I-ordered, a variable containing the I-index values is created, numbered **V1**, and called "**I**". For IJ-ordered data, two variables, numbered **V1** and **V2** and called "**I**" and "**J**," are created to contain the I- and J-index values. For IJK-ordered data, three variables "**I**", "**J**", and "**K**" are created and numbered **V1**, **V2**, and **V3**. The variable in the data file is numbered with the next available variable number, that is, **V2** for I-ordered data, **V3** for IJ-ordered data, and **V4** for IJK-ordered data. The created variables are the default X-, Y-, and Z-variables. The data type for the created variables is determined according to the following table:

| Maximum of IMax, JMax, and KMax | Data Type |
|---|---|
| < 256 | **BYTE** |
| <32,766 | **SHORTINT** |
| >=32,766 | **SINGLE** |

For example, if you have an ASCII file with 256 by 384 numbers representing intensities of a rasterized image, you could make a data file similar to the following:

```
VARIABLES = "TEMPERATURE"
 ZONE I=256, J=384
```
*List all 98,304 values of temperature here.*

Read the data file into Tecplot. Two new variables of type **SHORTINT** are created and used as the default X- and Y-coordinates. These variables are the I- and J-index values; they are named "**I**" and "**J**." You can now create any type of 2-D plot with the data.

If you have finite-element data, Tecplot will not create any new variables for you. If you need to add variables to finite-element data, you can do so using the Data menu.

## 5.3. Finite-Element Data

For finite-element data, the numerical values in the zone data must be in either **FEPOINT** or **FEBLOCK** format as specified by the **F** (*format*) parameter. The number of nodes (data points) is given by the **N=***numnodes* parameter, and the number of elements is given by the **E=***numelements* parameter (this is also the total length of the connectivity list). The element type (triangle, quadrilateral, tetrahedron or brick) is specified using the **ET** parameter. The zone data is divided into two logical sections (without any markers). The first section, the node data, lists the values of the variables at the data points (or nodes) as if they were I-ordered (one-dimensional) zone data. The second section, the connectivity list, defines how the nodes are

connected to form elements. There must be *numelements* lines in the second section; each line defines one element. The number of nodes per line in the connectivity list depends on the element type specified in the zone control line (**ET** parameter). (You may place blank lines between the node data and the connectivity list to help distinguish them.)

In the descriptions below, **N***E* is the *E*th node at a vertex of an element. The subscripts of **N***E* refer to the element number. For example, $\mathbf{N2}_3$ represents the second node of the third element.

For the triangle element type, each line of the connectivity list contains three node numbers that define a triangular element:

$\mathbf{N1}_M$, $\mathbf{N2}_M$, $\mathbf{N3}_M$

For the quadrilateral element type, each line of the connectivity list contains four node numbers that define a quadrilateral element:

$\mathbf{N1}_M$, $\mathbf{N2}_M$, $\mathbf{N3}_M$, $\mathbf{N4}_M$

If you need to mix quadrilateral and triangle elements, either create two zones or use the quadrilateral element type with node numbers ($\mathbf{N4}_M$=$\mathbf{N3}_M$) repeated to form triangles.

Zones created from the quadrilateral and triangle element types are called FE-surface zones.

For the tetrahedron element type, each line of the second section of the zone data contains four node numbers that define a tetrahedral element:

$\mathbf{N1}_M$ $\mathbf{N2}_M$ $\mathbf{N3}_M$ $\mathbf{N4}_M$

For the brick element type, each line of the second section contains eight node numbers that define a ‘‘brick-like” element:

$\mathbf{N1}_M$ $\mathbf{N2}_M$ $\mathbf{N3}_M$ $\mathbf{N4}_M$ $\mathbf{N5}_M$ $\mathbf{N6}_M$ $\mathbf{N7}_M$ $\mathbf{N8}_M$

Tecplot divides the eight nodes into two groups of four; nodes $\mathbf{N1}_M$, $\mathbf{N2}_M$, $\mathbf{N3}_M$, and $\mathbf{N4}_M$ make up the first group, and $\mathbf{N5}_M$, $\mathbf{N6}_M$, $\mathbf{N7}_M$, and $\mathbf{N8}_M$ make up the second group. Each node is connected to two nodes within its group and the node in the corresponding position in the other group. For example, $\mathbf{N1}_M$ is connected to $\mathbf{N2}_M$ and $\mathbf{N4}_M$ in its own group, and to $\mathbf{N5}_M$ in the second group. To create elements with fewer than eight nodes, repeat nodes as necessary, keeping in mind the basic brick connectivity just described. Figure 5-7 shows the basic brick connectivity. For example, to create a tetrahedron, you can set $\mathbf{N3}_M$=$\mathbf{N4}_M$ and $\mathbf{N5}_M$=$\mathbf{N6}_M$=$\mathbf{N7}_M$=$\mathbf{N8}_M$. To create a quadrilateral-based pyramid, you can set $\mathbf{N5}_M$=$\mathbf{N6}_M$=$\mathbf{N7}_M$=$\mathbf{N8}_M$. If you need a mixture of bricks and tetrahedra, either use two zones or use the brick element type with node numbers repeated so that tetrahedra result.

Zones created from the brick and tetrahedron element types are called FE-volume zones.

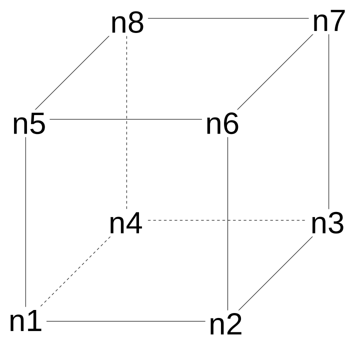



**Figure 5-7.** Basic brick connectivity.

If the keyword "**FECONNECT**" is specified in the **D** parameter in the zone control line, the connectivity list is duplicated from the previous zone. In this case, no connectivity list is given, just the node data. If you use **FECONNECT** for the first finite-element zone, Tecplot generates an error message.

## 5.3.1. Example of Triangle Data in FEPOINT Format

A simple example of triangle element type finite-element data in **FEPOINT** format is listed below. There are two variables (**X**, **Y**) and five data points. In this example, each row of the data section corresponds to a node and each column to a variable. Each row of the connectivity list corresponds to a triangular element and each column specifies a node number. This data set is plotted in Figure 5-8. Each data point is labeled with its node number.

```
VARIABLES = "X", "Y"
ZONE N=5, E=3, F=FEPOINT, ET=TRIANGLE
1.0 1.0
2.0 3.0
2.5 1.0
3.5 5.0
4.0 1.0

1 2 3
3 2 4
3 5 4
```

**5.3.1.1. The Same Data File in FEBLOCK Format.** The same data in **FEBLOCK** format is shown below. In this example, each column of the data section corresponds to a node

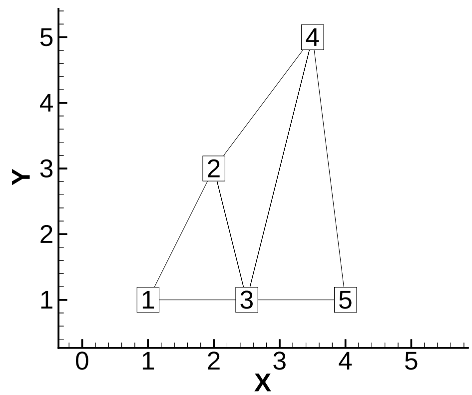



**Figure 5-8.** A finite-element triangle data set.

and each row to a variable. As above, each row of the connectivity list corresponds to a triangular element and each column specifies a node number.

```
VARIABLES = "X", "Y"
 ZONE N=5, E=3, F=FEBLOCK, ET=TRIANGLE
 1.0 2.0 2.5 3.5 4.0
 1.0 3.0 1.0 5.0 1.0
 1 2 3
 3 2 4
 3 5 4
```

## 5.3.2. An Example of FORTRAN Code to Generate Triangle Data in FEPOINT Format

The following sample FORTRAN code shows how to create triangle element type finite-element data in **FEPOINT** format:

```
 INTEGER VAR
   .
   .
   .
 WRITE (*,*) ´ZONE F=FEPOINT,ET=TRIANGLE,N=´, NNODES,´,E=´,NELEM
 DO 1 N=1,NNODES
     DO 1 VAR=1,NUMVAR
1        WRITE(*,*) VARRAY(VAR,N)

 DO 2 M=1,NELEM
```

```
      DO 2 L=1,3
2         WRITE (*,*) NDCNCT(M,L)
```

### 5.3.3. An Example of FORTRAN Code to Generate Triangle Data in FEBLOCK Format

The following sample FORTRAN code shows how to create triangle element type finite-element data in **FEBLOCK** format:

```
  INTEGER VAR
    .
    .
    .
  WRITE (*,*) ´ZONE F=FEBLOCK,ET=TRIANGLE,N=´,NNODES, ´,E=´,NELEM
  DO 1 VAR=1,NUMVAR
      DO 1 N=1,NNODES
1         WRITE(*,*) VARRAY(VAR,N)
  DO 2 M=1,NELEM
      DO 2 L=1,3
2         WRITE (*,*) NDCNCT(M,L)
```

### 5.3.4. An Example of a Finite-Element Zone Node Variable Parameters

The node variable parameter allows you to set the connectivity to match the value of the selected node variable. In the example below, the files appear to be identical in Tecplot, although the connectivity list has changed to reflect the values of the node variable Node Order. Notice that the index value of the nodes is not changed by the node variable value.

The original data set:

```
TITLE     = "Internally created dataset"
VARIABLES = "X"
"Y"
ZONE T="Triangulation"
 N=6, E=5,F=FEPOINT ET=Triangle
DT=(SINGLE SINGLE )
 2.00E+000 3.00E+000
 2.20E+000 3.10E+000
 3.10E+000 4.20E+000
 2.80E+000 3.50E+000
 2.40E+000 2.10E+000
 4.30E+000 3.20E+000
 1 2 5
```

```
6 4 3
5 4 6
2 3 4
5 2 4
```

The data set with the nodes re-ordered for connectivity:

```
TITLE     = "RE-ordered data"
VARIABLES = "X"
"Y" "Node-Order"
ZONE T="Triangulation"
 N=6, NV = 3, E=5,F=FEPOINT ET=Triangle
DT=(SINGLE SINGLE )
 2.00E+000 3.00E+000 5
 2.20E+000 3.10E+000 4
 3.10E+000 4.20E+000 1
 2.80E+000 3.50E+000 2
 2.40E+000 2.10E+000 6
 4.30E+000 3.20E+000 3
 5 4 6
 3 2 1
 6 2 3
 4 1 2
 6 4 2
```

## 5.4. Duplicating Variables and Connectivity Lists

The **`D`** parameter in the **`ZONE`** record allows you to duplicate variables or the connectivity list from the previous zone. The following is an example to illustrate this feature.

The table below shows Cartesian coordinates X and Y of six locations, and the pressure measured there at three different times (*P1*, *P2*, *P3*). The XY-locations have been arranged into finite-elements.

| X | Y | $P_1$ | $P_2$ | $P_3$ |
|---|---|---|---|---|
| -1.0 | 0.0 | 100 | 110 | 120 |
| 0.0 | 0.0 | 125 | 135 | 145 |
| 1.0 | 0.0 | 150 | 160 | 180 |
| -0.5 | 0.8 | 150 | 165 | 175 |
| 0.5 | 0.8 | 175 | 185 | 195 |
| 0.0 | 1.6 | 200 | 200 | 200 |

For this case, we want to set up three zones in the data file, one for each time measurement. Each zone has three variables: X, Y, and P. The zones are of the triangle element type, meaning that three nodes must be used to define each element. One way to set up this data file would be to list the complete set of values for X, Y, and P for each zone. Since the X,Y-coordinates are exactly the same for all three zones, a more compact data file can be made by using the duplication list parameter (*D*). In the data file given below, the second and third zones have duplication lists that copy the values of the X- and Y-variables and the connectivity list from the first zone. As a result, the only values listed for the second and third zones are the pressure variable values. A plot of the data is shown in Figure 5-9. Note that the data could easily have been organized in a single zone with five variables. Since blank lines are ignored in the data file, you can embed them to improve readability.

```
TITLE = "Example: Duplicated Variables and Connectivity Lists"
VARIABLES = "X", "Y", "P"
ZONE T="P_1", F=FEPOINT, N=6, E=4, ET=TRIANGLE
-1.0  0.0  100
0.0  0.0  125
1.0  0.0  150
-0.5  0.8  150
0.5  0.8  175
0.0  1.6  200

1 2 4
2 5 4
3 5 2
5 6 4

ZONE T="P_2", F=FEPOINT, N=6, E=4, ET=TRIANGLE, D=(1,2,FECONNECT)
110 135 160 165 185 200

ZONE T="P_3", F=FEPOINT, N=6, E=4, ET=TRIANGLE, D=(1,2,FECONNECT)
120 145 180 175 195 200
```

## 5.5. Converting ASCII Data Files to Binary

Although Tecplot can read and write either ASCII or binary data files, binary data files are more compact and are read into Tecplot much more quickly than ASCII files. Your Tecplot distribution includes a program called Preplot that converts ASCII data files to binary data files. You can also use Preplot to debug ASCII data files that Tecplot cannot read.

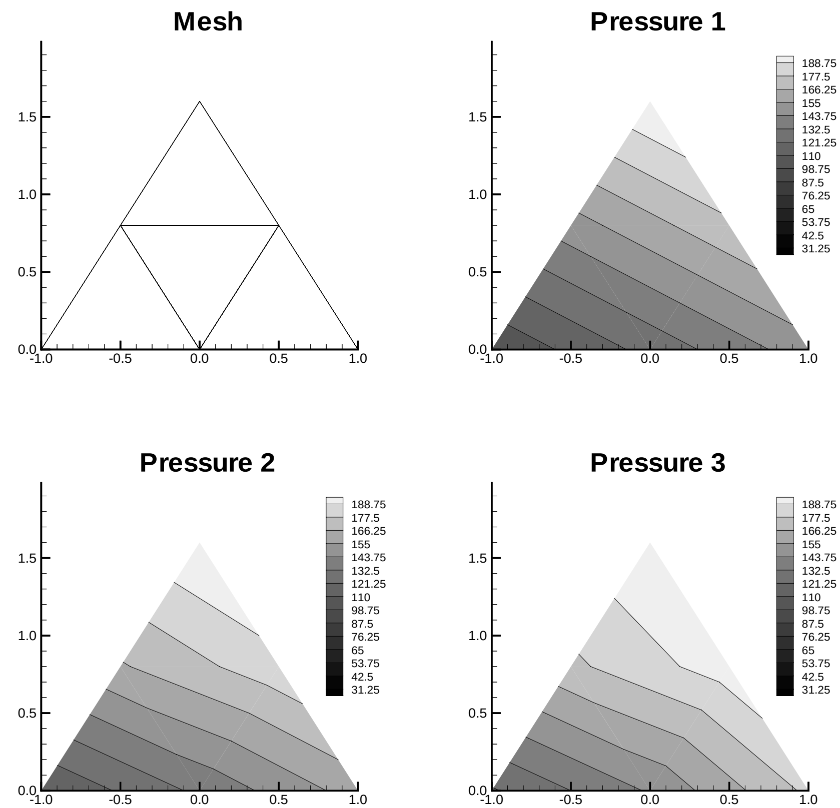



**Figure 5-9.** Plot of finite-element zones.

### 5.5.1. Standard Preplot Options

To use Preplot, type the following command from the UNIX shell prompt, from a DOS prompt, or using the Run command in Windows:

**`preplot`** *infile [outfile] [options]*

where *infile* is the name of the ASCII data file, *outfile* is an optional name for the binary data file created by Preplot, and *options* is a set of options from either the standard set of Preplot options or from a special set of options for reading PLOT3D format files. If *outfile* is not spec-

ified, the binary data file has the same base name as the *infile* with a "**.plt**" extension. You may use a minus sign ("**-**") in place of either the *infile* or *outfile* to specify "standard input" or "standard output," respectively.

Any or all of **-iset**, **-jset**, and **-kset** can be set for each zone, but only one of each per zone.

For more Preplot command lines, see Appendix B.3, "Preplot."

### 5.5.2. Examples of Using Preplot

If you have an ASCII file named **dset.dat**, you can create a binary data file called **dset.plt** with the following Preplot command:

```
preplot dset.dat dset.plt
```

By default, Preplot looks for files with the **.dat** extension, and creates binary files with the **.plt** extension. Thus, either of the following commands is equivalent to the above command:

```
preplot dset
```

```
preplot dset.dat
```

Preplot checks the input ASCII data file for errors such as illegal format, numbers too small or too large, the wrong number of values in a data block, and illegal finite-element node numbers. If Preplot finds an error, it issues a message displaying the line and column where the error was first noticed. This is only an indication of where the error was *detected*; the actual error may be in the preceding columns or lines.

If Preplot encounters an error, you may want to set the debug option to get more information about the events leading up to the error:

```
preplot dset.dat -d
```

You can set the flag to **-d2**, or **-d3**, or **-d4**, and so forth, to obtain even more detailed information.

In the following Preplot command line, the number of points that are written to the binary data file **dset.plt** is less than the number of points in the input file **dset.dat**:

```
preplot dset.dat -iset 3,6,34,2 -jset 3,1,21,1 -iset 4,4,44,5
```

For zone 3, Preplot outputs data points with I-index starting at 6 and ending at 34, skipping every other one, and J-index starting at one and ending at 21. For zone 4, Preplot outputs data points with the I-index starting at four, ending at 44, and skipping by five.

In the following Preplot command line, every other point in the I-, J-, and K-directions is written to the binary data file:

```
preplot dset.dat -iset ,,,2 -jset ,,,2 -kset ,,,2
```

The *zone*, *start*, and *end* parameters are not specified, so all zones are used, starting with index 1, and ending with the maximum index. The overall effect is to reduce the number of a data points by a factor of about eight.

### 5.5.3. Using Preplot to Convert Files in PLOT3D Format

PLOT3D is a graphics plotting package developed at NASA. Some numerical simulation packages and other programs can create graphics in PLOT3D format. There are two paths by which you can get files in PLOT3D format into Tecplot. This section describes the Preplot path; you can also use the PLOT3D loader described in Section 7.9, "The PLOT3D Data Loader."

Preplot can read files in the PLOT3D format and convert them to Tecplot binary data files through the use of special switches. You do not need to know about these switches unless you have data in PLOT3D format.

PLOT3D files typically come in pairs consisting of a grid file (with extension **`.g`**) and a solution file (with extension **`.q`**). Sometimes only the grid file is available. The grid itself may be either a single grid, or a multigrid, and the data may be 1D, 2D, 3D-planar, or 3D-whole (equivalent to Tecplot's 3-D volume data). The PLOT3D files may be binary or ASCII. The PLOT3D-specific switches to Preplot allow you to read PLOT3D files with virtually any combination of these options.

The *ilist*, *jlist*, and *klist* are comma-separated lists of items of the form:

*start*[:*end*][:*skip*]]

where *start* is the number of the starting I-, J-, or K-plane, *end* is the number of ending I-, J-, or K-plane, and *skip* is the skip factor between planes. If *end* is omitted, it defaults to the starting plane (so if just *start* is specified, only that one plane is included). The *skip* defaults to one (every plane) if omitted; a value of two includes every other plane, a value of three include every third plane, and so on.

You must specify one of the flags **`-1d`**, **`-2d`**, **`-3dp`**, or **`-3dw`**. You may also specify only one of **`-ip`**, **`-jp`**, or **`-kp`** and only one of **`-b`** or **`-f`**.

If the input PLOT3D file is 3-D whole (**`-3dw`**) and none of the plane-extraction switches **`-ip`**, **`-jp`**, or **`-kp`** is specified, the PLOT3D file is converted directly to an IJK-ordered zone (or multiple zones if the file is multi-grid).

For example, in the following command line, Preplot reads from the PLOT3D files **`aero.g`** and **`aero.q`**. The input is binary and 3-D whole. The J-planes 2, 3, 4, 45, 46, and 47 are processed and made into six IJ-ordered zones, in a binary data file named **`aero.plt`**:

```
preplot aero -plot3d -b -3dw -jp 2,3,4,45,46,47
```

In the following command line, the plane-extraction switches are omitted, so Preplot creates a single IJK-ordered zone:

```
preplot aero -plot3d -b -3dw
```

The following command line reads an ASCII file **`airplane.g`** for which there is no corresponding **`.q`** file; the data is 3-D whole:

```
preplot airplane -plot3d -gridonly -3dw
```

The following command line reads a multi-grid, 3-D planar, binary-FORTRAN pair of PLOT3D files, **`multgrid.g`** and **`multgrid.q`**:

```
preplot multgrid -plot3d -m -f -3dp
```

# CHAPTER 6 Working with Tecplot Files

Tecplot reads and writes a number of different files. In general, file operations are performed in the File menu. Using this menu, you can instruct Tecplot to read or write data files, layout files, and macro files. Some files, such as stylesheets and equation files, are read and written from other menus and dialogs.

This chapter presumes that your data files are in a form readable by Tecplot. Chapter 5, "Formatting ASCII Data for Tecplot," discusses the format for data files structured for direct reading by Tecplot. Chapter 7, "Data Loaders: Tecplot's Import Feature," discusses the importation of a variety of other data formats.

Some files can be loaded from the Tecplot command line. For more information, please see Appendix A, "Tecplot Command Line Options."

## 6.1. Loading Tecplot-Format Data Files

Once your data files are structured for Tecplot, loading them is the next step. This section describes the process for Tecplot-format data files and for reading data files formatted for other software packages.

Tecplot cares about the format of a data file and the context in which the file is read, but is not concerned about file names (including extensions). Tecplot uses standard extensions (**`.dat`** for ASCII, **`.plt`** for binary).

Tecplot can read and write both ASCII and binary Tecplot-format data files. Binary files are generally more compact and efficient. ASCII files are "human-readable," but they are larger than binary files, and take longer to load. In general, if you have a large amount of data, you will want to use binary files.

There are four ways to work with Tecplot-format data files:

- Generate an ASCII Tecplot-format data file. Read the ASCII data file into Tecplot and work with it without any conversion. If the data set is altered, save it as an ASCII data file. This method works for smaller data sets where the convenience of an ASCII file outweighs the inefficiencies.
- Generate an ASCII Tecplot-format data file. When you first work with a data file, read it into Tecplot, then save it as a binary data file. From then on, work with the binary file. Once you have saved a binary version of the data file, you can delete the ASCII version to save space. This method works well for large data sets where the inefficiencies of ASCII format are noticeable.
- Generate an ASCII Tecplot-format data file, and then convert the file to binary with Preplot. Preplot is a utility program included with your Tecplot distribution which converts ASCII Tecplot-format data files to binary Tecplot-format data files (Preplot can also convert PLOT3D files to Tecplot-format binary data files). Once the binary file is created, you can delete the ASCII version to save space. This method works well for identifying problems with data files, since Preplot's error messages can be more detailed than Tecplot's. This method also works well in batch processing, or if the ASCII data files are generated on another machine. See Section 5.5, "Converting ASCII Data Files to Binary," for a description of Preplot.
- Generate a binary Tecplot-format data file. Read the binary data file into Tecplot and work with it without any conversion. This method works well if you can use the routines provided by Amtec that write Tecplot-format binary files from C or FORTRAN programs. See Chapter 11, "Writing Binary Data for Loading into Tecplot," in the *Tecplot Reference Manual*, for complete details.

### 6.1.1. Loading Data Files

Tecplot allows you to read multiple data files to create a data set, but most often, you will start with a single data file. Tecplot uses separate procedures for loading single versus multiple data files.

To load a single data file:

1. From the File menu, choose Load Data File(s). The Load Data File dialog appears, as shown in Figure 6-1.
2. Enter the file name in the File Name field and click Open. You may have to include the file path or use the Look in drop-down to navigate to the file's folder.

To load multiple data files:

1. From the File menu, choose Load Data File(s). The Load Data File dialog appears.
2. Click Multiple Files. The Load Data File dialog for multiple files appears, as shown in Figure 6-1.

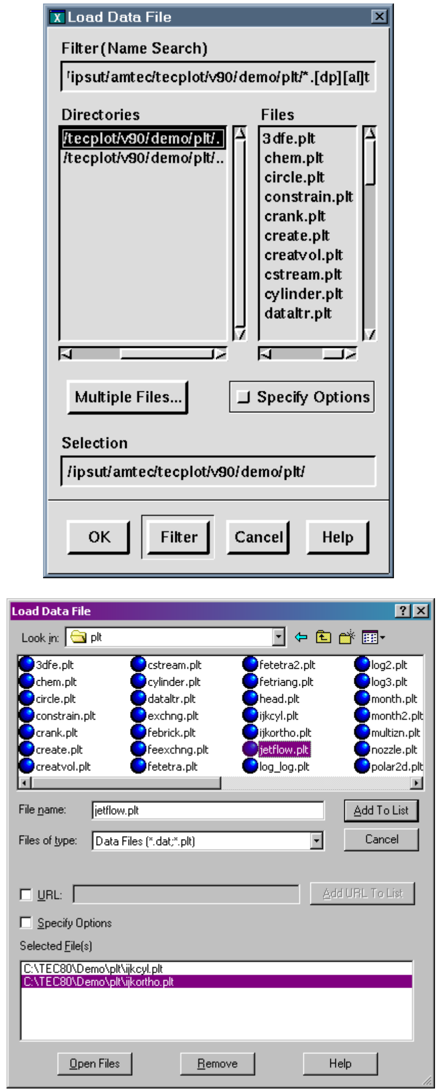



**Figure 6-1.** Load Data File dialog in Motif (top) and the Load Multiple Data Files dialog in Windows (bottom).

3. Select a file following the procedure in step 2 above regarding how to load a single data file. Open will now read Add To List. Click on it to add the file, repeat this process for any remaining files you wish to add. You may specify a URL by selecting the URL check box

and entering a URL. The URL must start with ftp:// or http://. For example, ftp://ftp.microsoft.com/myplot.plt. If you wish to add as disk file name you must deselect the URL check box. You may add any combination of URLs and disk file names to the Selected File(s) list.

or

Double-click on the name of the file in the Look in drop-down list. The file is added to the Selected File(s) scrolled list.

4. To remove files from the Selected File(s) scrolled list, select the files, then click Remove.
5. When the Selected File(s) scrolled list contains only those files you want to read in, click Open Files to read in the files.

The order of the list of files to be read in is important. Tecplot numbers the zones based on the ordering in this list.

**6.1.1.1. Options when Loading Data Files.** Tecplot allows you to specifically control what is loaded from your data files. This gives you the ability to load only certain zones or variables, or even part of a zone. You can also choose to load the variables in your data files by name or by position. The Load Data File Options dialog is where you can specify these and other options.

To select options when loading one or more data files:

1. Select files for loading as described in Section 6.1.1, “Loading Data Files,” but do not click Open.
2. Select the Specify Options check box.
3. Click Open. The Load Data File Options dialog appears, as shown in Figures 5-2 through 5-5.
4. Select your desired options. (See below for more information about all of the options available.)
5. Click OK to close the Load Data File Options dialog and read the selected data.

The Load Data File Options dialog is a multi-page dialog. Clicking each button or tab at the top of the dialog will show you a different page or group of options (however, when clicking OK all of the options will be used). The Load Data File Options dialog has three pages—General, Zones, and Variables—which are discussed in Sections 6.1.1.2. through 6.1.1.6.

**6.1.1.2. General Options when Loading Data Files.** On the General page (Figure 6-2), you have the option to load a subset of record types, to load only portions of the data, and to specify in which frame mode the data will initially be displayed.

If you want to load only specific record types from the data file, you can choose the desired record types by selecting the appropriate check boxes, as follows:

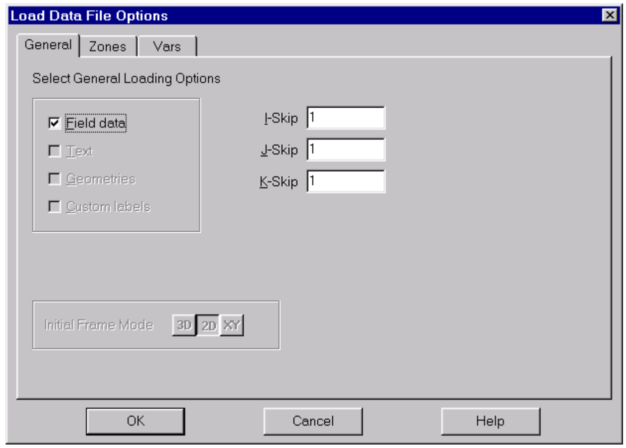



**Figure 6-2.** The General page of the Load Data File Options dialog.

| | |
|---|---|
| Field data | Load zone records (these are the actual data). If this option is not selected, the Zones and Variables pages of the dialog will be gray, as they allow you to specify more details about the data to be loaded. |
| Text | Load text records. |
| Geometries | Load geometry records. |
| Custom labels | Load custom label records. |

The check boxes will be available only if those records exist in the data files you selected to load. By default, all of the records which exist in the data files are selected.

If you want to load only a portion of the data points, you can specify skip factors for the I-, J-, and K-dimensions in the I-, J-, and K-Skip text fields. Each skip factor *n* tells Tecplot to read in every *n*th point in the specified direction. By default, all the skip factors are set to 1, so that every data point is loaded.

If you want to specify the initial frame mode for the data, you can select one of the Initial Frame Mode buttons. By default, I-ordered data has an initial frame mode of XY, IJ-ordered and finite-element surface data has an initial frame mode of 2D, and IJK-ordered and finite-element volume data have an initial frame mode of 3D.

**6.1.1.3. Zone Options when Loading Data Files.** On the Zones page, shown in Figure 6-3, you have the option of selecting specific zones to load from the data files and, if appropriate, whether to “collapse” the zone list.

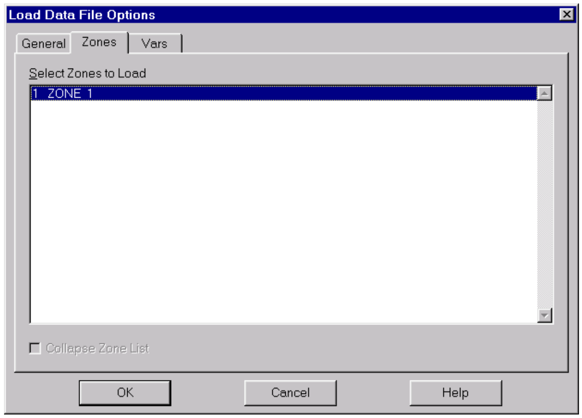



**Figure 6-3.** The Zones page of the Load Data File Options dialog.

If you want to specify which zones to load from the data files, you can select them in the Select Zones to Load list. This is a multiple selection list so you can either click and drag, Ctrl-click, or Shift-click to get the set of zones which you want to load. The zones are listed in the order in which the data files were chosen (that is, all of the zones from the first data file are listed, then all of the zones from the second data file, and so forth). By default, all of the zones are selected to be loaded.

If you have selected to only load specific zones and want them renumbered when they are read in, select the Collapse Zone List check box (If you are loading variables by position, the check box will be labeled Collapse Zone and Variable Lists). See Section 6.1.1.7, “Collapsing Zone and Variable Lists,” for more information.

**6.1.1.4. Variable Options when Loading Data Files.** On the Variables page, you have the option of loading your variables By Name or By Position (as shown in Figure 6-4 and Figure 6-5). Select to load your variables by name by selecting the By Name option and by position by selecting the By Position option. The default is to load variables by position. If you are loading variables by position, then none of the options you see when By Name is selected have any effect, and vice-versa.

When loading variables by position, Tecplot variables are created based on the order in which variables are listed in the data files. Tecplot’s first variable is created from the first variable in each data file, then Tecplot’s second variable is created from the second variable in each data file, and so on.

When loading variables by name, Tecplot initially tries to create variables based on the variable names in the data files. Tecplot’s first variable is created from the first variable name common

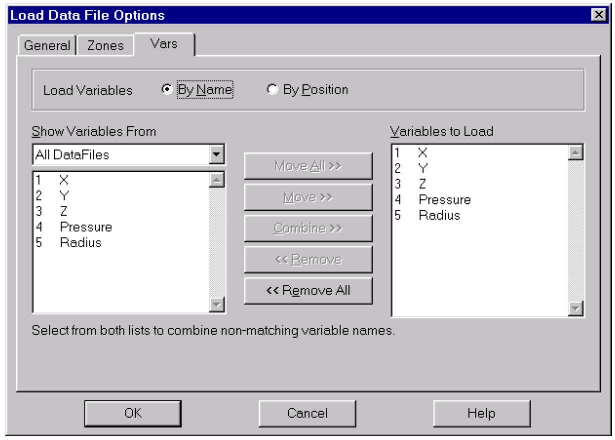



**Figure 6-4.** The Variables page of the Load Data File Options dialog (Load Variables by Name option).

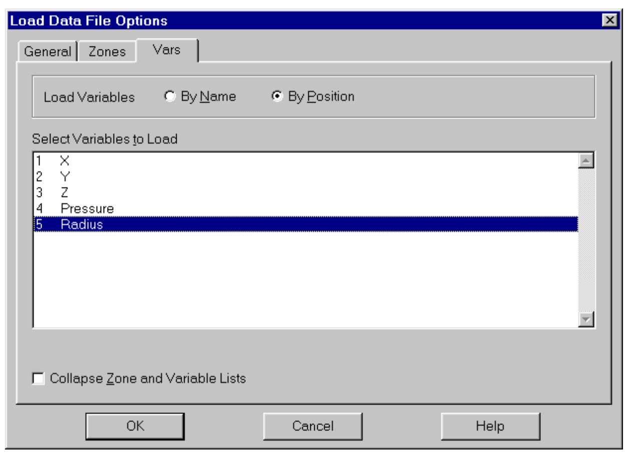



**Figure 6-5.** The Variables page of the Load Data File Options dialog (Load Variables by Position option).

to all data files, then Tecplot's second variable is created from the second variable name common to all data files, and so on.

In the data set it creates, Tecplot sets variable order based on the order of variables in the first data file it loads. This is true both when loading by position and by name.

To see the difference between loading by position and loading by name, consider a hypothetical example where two data files are being loaded: **`file1.plt`** and **`file2.plt`**. File

**`file1.plt`** has variables **X**, **Y**, and **P**, in that order. File **`file2.plt`** has variables **P**, **X**, and **Y**, in that order.

| Data File | `file1.plt` | `file2.plt` |
|---|---|---|
| **Variable 1** | **X** | **P** |
| **Variable 2** | **Y** | **X** |
| **Variable 3** | **P** | **Y** |

If these two files were loaded by position, the first variable in Tecplot would be created from **X** in **`file1.plt`** and **P** in **`file2.plt`**. The second variable in Tecplot would be created from **Y** in **`file1.plt`** and **X** in **`file2.plt`**. The third variable in Tecplot would be created from **P** in **`file1.plt`** and **Y** in **`file2.plt`**. In other words, loading variables by position would load the first variable from both files, then the second variable from both files, then the third.

If these two files were loaded **by name**, the first variable in Tecplot would be created from **X** in **`file1.plt`** and **X** in **`file2.plt`**. The second variable in Tecplot would be created from **Y** in **`file1.plt`** and **Y** in **`file2.plt`**. The third variable in Tecplot would be created from **P** in **`file1.plt`** and **P** in **`file2.plt`**. In other words, loading variables by name would load the first variable from **`file1.plt`** and the second variable from **`file2.plt`**; then, the second variable from **`file1.plt`** and the third variable from **`file2.plt`**; then, the third variable from **`file1.plt`** and the first variable from **`file2.plt`**.

When appending data files to the current data set, or replacing the current data set but retaining the plot style, you will not be able to select whether to load the variables by name or by position. The choice is fixed based on the current data set in Tecplot.

**6.1.1.5. Loading Variables by Name.** The Variables page when loading variables by name is shown in Figure 6-5. You have the option of selecting specific variable names to load from the data files.

When loading variables by name, the variables are associated by name and then loaded into Tecplot. Variable names can be combined so that a variable in one data file with a different name in another data file can be loaded into one Tecplot variable. If a variable name is missing in a given file, that Tecplot variable will be set to all zeros for all zones loaded from that file.

When loading variables by name from the Variables page, there are two lists of variables with several options between them.

The list on the left (the Show Variables From list) shows the variable names from the data files which you selected to load. You can filter this list with the drop-down at the top of the list, which allows you to select any of the data files you are loading, or select All Data Files. The

All Data Files option shows the variable names from all of the data files, listed in order (that is, all of the variable names from the first data file are listed, then all of the variable names from the second file, and so forth). However, if a variable name exists in more than one data file, it only appears once in this list. An asterisk (`*`) next to a variable name in this list indicates that the variable name does not exist in all data files which are being loaded. A number next to a variable name in this list indicates that the variable name will be loaded into that Tecplot variable. For example, if a "2" appears next to a variable name in this list, it means that the variable name will be loaded into the second Tecplot variable.

The list on the right (the Variables to Load list) shows the variables which will be loaded into Tecplot when you click OK. By default, this list shows only variable names which exist in all of the data files you selected to load. So, if no matching variable names exist, this list will initially be empty. You will get an error if you click OK while the Variables to Load list is empty. An asterisk (`*`) next to a variable name in this list indicates that the variable name does not exist for all data files which are being loaded. If you click OK with an asterisk in this list, the files which do not contain the variable name will have their zones set to zero for that variable. No duplicate variable names are allowed in this list.

The options separating the two lists allow you to manipulate the lists by moving variable names between the two lists. Some perform operations based on items that are selected in the lists. If no appropriate items are selected, these will be inactive.

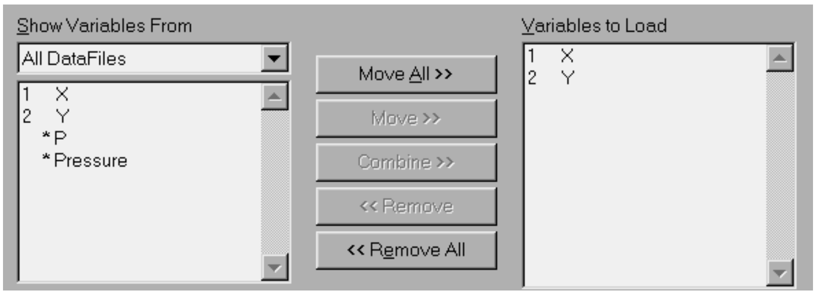
- The [Move All >>] button takes all of the variable names listed in the Show Variables From list and adds them to the Variables to Load list.
- The [Move >>] button takes the variable name which is selected in the Show Variables From list and adds it to the Variables to Load list.
- The [Combine >>] button takes the variable name which is selected in the Show Variables From list and combines it with the variable name which is selected in the Variables to Load list.
- The [<< Remove] button removes the selected variable name from the Variables to Load list.
- The [<< Remove All] button removes all of the variable names from the Variables to Load list.

For example, look at the case where two data files are being loaded: **`file1.plt`** and **`file2.plt`**. File **`file1.plt`** has variables **`X`**, **`Y`**, and **`P`**. File **`file2.plt`** has variables **`X`**, **`Y`**, and **`Pressure`**.

| **Data File** | **`file1.plt`** | **`file2.plt`** |
|---|---|---|
| Variable 1 | **`X`** | **`X`** |

| | | |
|---|---|---|
| Variable 2 | **Y** | **Y** |
| Variable 3 | **P** | **Pressure** |

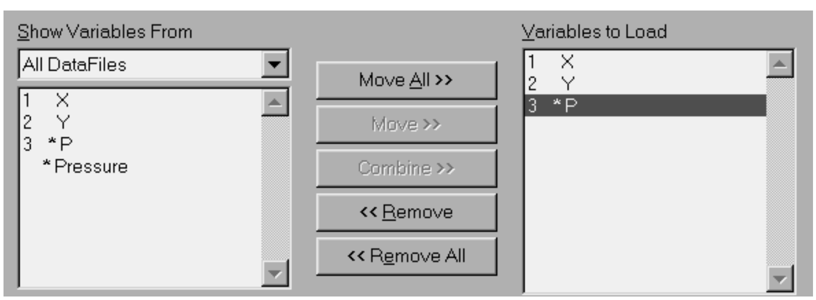
If you loaded these two files and selected to go to the Load Data File Options dialog on the By Name Variables page, you would see the variable names listed as follows:



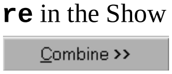

Note the numbers 1 and 2 listed next to the variable names in the Show Variables From list. This indicates that these variables will be loaded into Tecplot in positions 1 and 2. Also note the asterisks (**`*`**) next to the **P** and **Pressure** variables. This indicates that these variable names do not exist in all files. In this case, **P** does not exist in **file2.plt** and **Pressure** does not exist in **file1.plt**. For this reason, the variables **P** and **Pressure** do not appear by default in the Variables to Load list. If you wanted to load these variables together as one Tecplot variable, you would first select the variable name **P** in the Show Variables From list.

Next, click the 

button. The dialog now appears as follows:

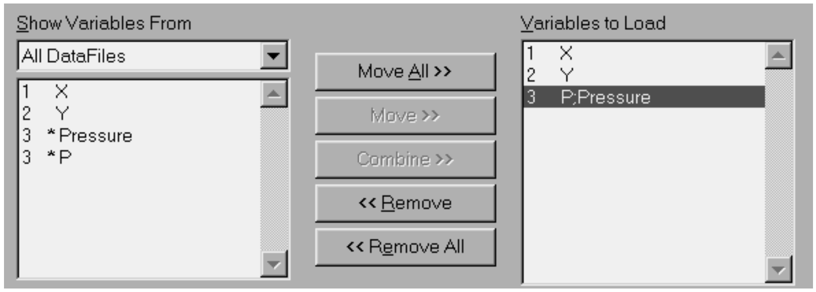



Note the asterisk (**`*`**) that appears next to **P** in the Variables to Load list. This again indicates that the variable name **P** does not exist in all files (in this case, **file2.plt**). If you were to click OK at this time the variable **P** would be filled with zeros for all zones which came from **file2.plt**. To remedy this, we would like to add the variable name **Pressure** to the variable name **P** to combine these two variable names into one Tecplot variable. First, select **Pressure** in the Show Variables From list. Next, select **P** in the Variables to Load list. Next,

click the 

button.

The dialog now looks like:




Note that there is now no asterisk (*) next to the third variable in the Variables to Load list. This indicates that either a variable name **P** or a variable name **Pressure** exists in all data files. Clicking OK at this time would load the first three variables from both data files.

When Tecplot encounters a combined variable name like **P;Pressure**, it looks at the first data file and tries to find a variable named **P**. If it finds one, it loads that variable. If it cannot, it then tries to find a variable named **Pressure**. If that too fails, then the variable for the zones from that file is filled with zeros.

When appending data files to the current data set or replacing the current data set but retaining the plot style, you will have limited options in changing the Variables to Load list. The list is partially fixed based on the current data set in Tecplot. You can add variable names to the list, and combine new variable names with the list, but you will not be able to remove any of the current variable names from the list.

**Note:** When appending data files to the current data set by adding variable names to the Variables to Load list, adding a new variable name which already exists in the current data set's data files but which was not initially loaded will cause Tecplot to reload the affected data files to include these new variable names. You must first change the data set by the Alter option from the Data menu, or by other means.

**6.1.1.6. Loading Variables by Position.** The Variables page when loading variables by position is shown in Figure 6-6. Here, you have the option of selecting specific variables to load from the data files, and if appropriate, whether to "collapse" the zone and variable lists.

If you want to specify which variables to load from the data files, you can select them in the Select Variables to Load list. This is a multiple selection list so you can either click-and-drag, Ctrl-click, or Shift-click to get the set of variables which you want to load. The variable names listed come from the first data file. If variable names in the other data files do not match those in the first data file, an asterisk (*) will appear next to the variable name. The number of variables listed is limited to the minimum number of variables across all of the data files. So, for example, if you are loading three files, one with five variables, one with six variables, and one

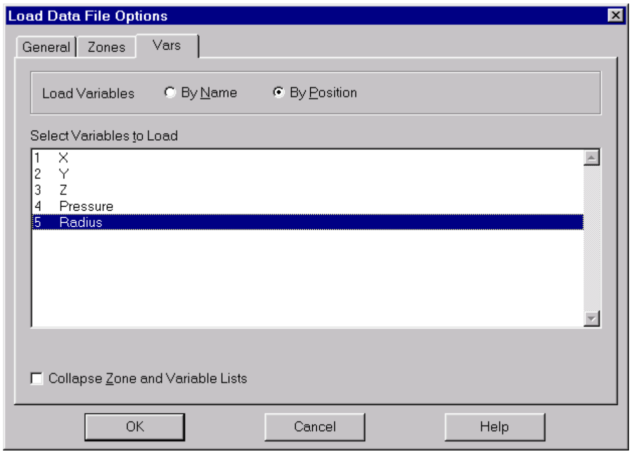



**Figure 6-6.** The Variables page, Load Variables By Position option, of the Load Data File Options dialog.

with four variables, you will only be able to load the first four variables out of each data file. By default, all of the variables are selected to be loaded.

If you have selected to only load specific variables and want them renumbered, select the "Collapse Zone and Variable Lists" check box. See Section 6.1.1.7, "Collapsing Zone and Variable Lists," for more information.

When appending data files to the current data set or replacing the current data set but retaining the plot style, you will not be able to select which variables to load. These are fixed to the variables currently in Tecplot. When appending data files to the current data set, the new data files must have at least as many variables as are currently in Tecplot.

**6.1.1.7. Collapsing Zone and Variable Lists.** When you load your data files into Tecplot, you have the option of reading only selected zones (and variables when loading them by position). When you perform such a partial read, you have the option of either preserving the existing zone and variable numbering or "collapsing" the data that is actually read so that zones and variables are renumbered according to their positions in Tecplot.

For example, suppose you have a data file with five zones, and you want to read only zones 2 and 5. If you choose not to collapse your zones and variables (the default), Tecplot reads in the zones as zones 2 and 5. If you then write this data set to a file in ASCII format, you will see that the data set does indeed have five zones; zones 1, 3, and 4 are zones with no data called "Zombie" zones. If you choose the collapse option, Tecplot reads in the zones as zones 1 and 2. Subsequently writing the data set to a file results in a file with only two zones.

Similarly, if you are loading variables by position and want to read variables 1, 5, and 6, the default is to preserve the variable numbering, so that you continue to work with variables 1, 5, and 6. If you write out the data set to a file in ASCII format, you will see nameless variables, with values of zero, where the unread variables would have been. If you choose the collapse option, Tecplot renumbers the variables as variables 1, 2, and 3.

If you are appending data files, the Collapse Zone List check box is set based on the current data set in Tecplot, and you cannot change it. The zones will be numbered offset from Tecplot's current zones. For example, if you partially read zones 1 and 5 from a data file and collapse the zone list, you will get zones 1 and 2. Then, when you append a data file with zones 2 and 4 you will get new zones numbered 3 and 4 in Tecplot. If instead, you had not collapsed the zone list in the first data file, you would have had zones 1 and 5, and when you appended zones 2 and 4 from the second data file you would get new zones 7 and 11 in Tecplot.

In most cases, you will not need to collapse zones and variables. All dialogs that show zones or variables will list the zones you read in; they just won't be numbered sequentially. There are certain situations when you definitely do not want to collapse your zones and variables:

- You have a large data set and read a portion of the data to reduce the amount of memory Tecplot requires to process the data. You then create a stylesheet that you want to use at a later time with a different subset of the data.
- You have many zones and variables and you are familiar with certain ranges of zones or variables. For example, you may know that zones 150-200 represent a known portion of the data. If you do a partial read and do not collapse the data, these zones will continue to be designated with their familiar numbers.

## 6.2. Writing Data Files

You can write out the data set in the current frame as either an ASCII or binary data file. Every time you write a file, you are given the opportunity to choose which part of the data to write, and to specify the format for the saved file.

To write the data set in the current frame to a file:

1. From the File menu, choose Write Data File. The Write Data File Options dialog appears, as shown in Figure 6-7.
2. (Optional) Select the zones and or variables you want to write to the saved data file.
3. (Optional) Specify which record types you want to write to the saved data file by selecting the appropriate check boxes:
   - **Text:** Select this check box to save any text attached to the current frame.
   - **Geom:** Select this check box to save any geometries attached to the current frame.

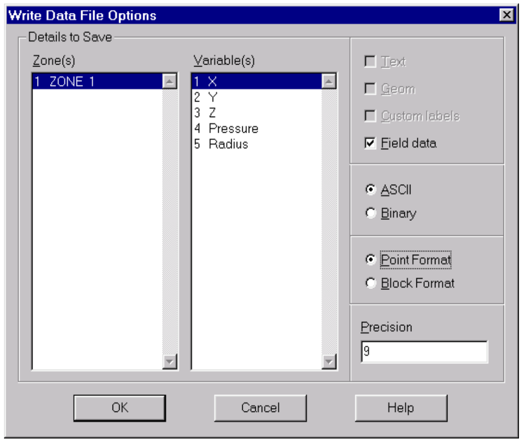



**Figure 6-7.** The Write Data File Options dialog.

- **Custom labels:** Select this check box to save any custom labels attached to the current data set.
- **Field data:** Select this check box to save the zone data.

By default, all record types that are present in the current data set are selected.

4. (Optional) Choose whether to save the file as ASCII or Binary by selecting the appropriate option button.
5. (Optional) If you choose ASCII, choose whether to write the file in **`POINT`** format or **`BLOCK`** format. (**`POINT`** format organizes the data by data points, **`BLOCK`** format organizes it by variables. See Chapter 5, “Formatting ASCII Data for Tecplot,” for a complete description of both formats.) You can also specify the precision of your Float and Double variables. These variable types are written in exponent format and the precision determines the number of digits included past the decimal point (a precision of three would result in numbers of the form **`4.657E+02`**).
6. Click OK to call up the Write Data File dialog shown in Figure 6-8.
7. Specify a file name.
8. Click OK to write the data. If you have selected the file name of an existing file, Tecplot prompts you for confirmation before overwriting the file.

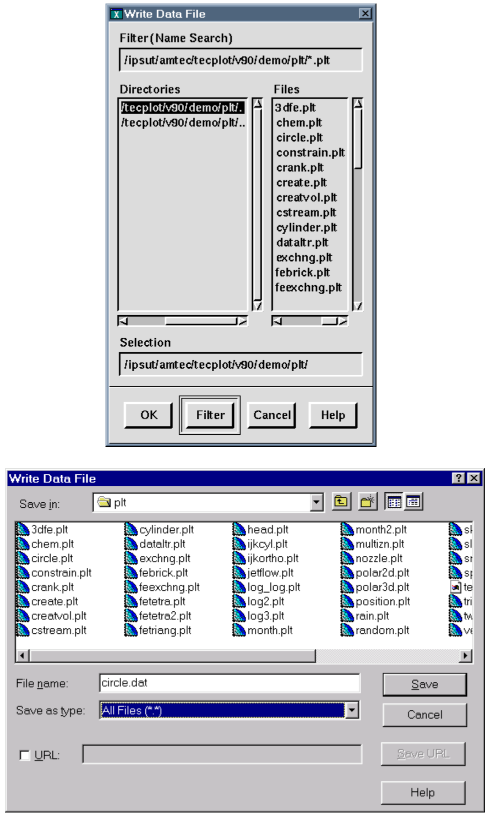



**Figure 6-8.** The Write Data File dialog in Motif (top) and Windows (bottom).

## 6.3. Layout Files, Layout Package Files and Stylesheets

Tecplot has three different types of files for storing plot information:

- **Stylesheets:** Stylesheets store information about a single frame, but do not include any information about the data used by the frame.
- **Layout files:** Layout files store information about all the frames in the workspace, including identification of the data used by each frame.
- **Layout Package files:** Layout package files are an extension to layout files where data and an optional preview image are included.

This section describes the three types of files and gives suggestions on when to use each one.

### 6.3.1. Stylesheets

Layout and layout package files for the most part are the preferred medium for saving the style of your plot. They are quick and easy to load and save, and they save your complete picture. Unlike layout and layout package files, which contain the complete style of all the frames in your workspace, Stylesheets contain only the style of a single frame in Tecplot. Despite this limitation, there are a few situations where Stylesheets are useful. These are:

- When pre-processing must be done to a data set prior to attaching a style. You may need to load a data set and run some equations or do interpolation or zone extraction before you can assign a style. The style may reference objects or variables that do not exist in the original data and it is necessary to assign the style after they are created.
- When switching styles on large data sets. You may want to load a large data set and generate two full page plots. Each plot has a different style. By using a stylesheet for the second plot you avoid having to reload the data set.
- When copying the style of one frame to another frame in the same layout.
- When saving just part of a frame's style, such as just the contour levels.

A stylesheet is a special type of macro file that contains commands used to define the style of a single frame in Tecplot. Figure 6-9 shows some of the items that are part of the style of a frame. The style of a frame includes such attributes as what type of plot is being drawn (a 2-D contour plot or an XY-plot) and what colors are being used. The style also defines the current view of the data and how the axes are drawn (if at all). Any enhancements to the frame, such as text or line art (geometries) are also part of the style of a frame.

Stylesheets are copied (created, by extracting from a frame and writing to a file) or pasted (applied to a frame) by using the Copy Style to File or Paste Style from File options in the Style menu. A stylesheet does not contain any information about the frame's data (if required), or where that data set comes from.

To select part or all of a frame style and create a stylesheet:

1. In the Style menu, click Copy Style to File. The Copy Style to File dialog appears.
2. Select a path and file name.
3. (Optional) Click Options. The Copy Style Options dialog will appear. In the Copy Style Options dialog, select the aspects of the frame style that you want to save, then click Close.
4. In the Copy Style to File dialog, click OK.

To apply all or part of a stylesheet to a data set:

1. In the Style menu, click Paste Style from File. The Paste Style from File dialog appears.
2. Specify the path and name of the file which has the style to be copied.

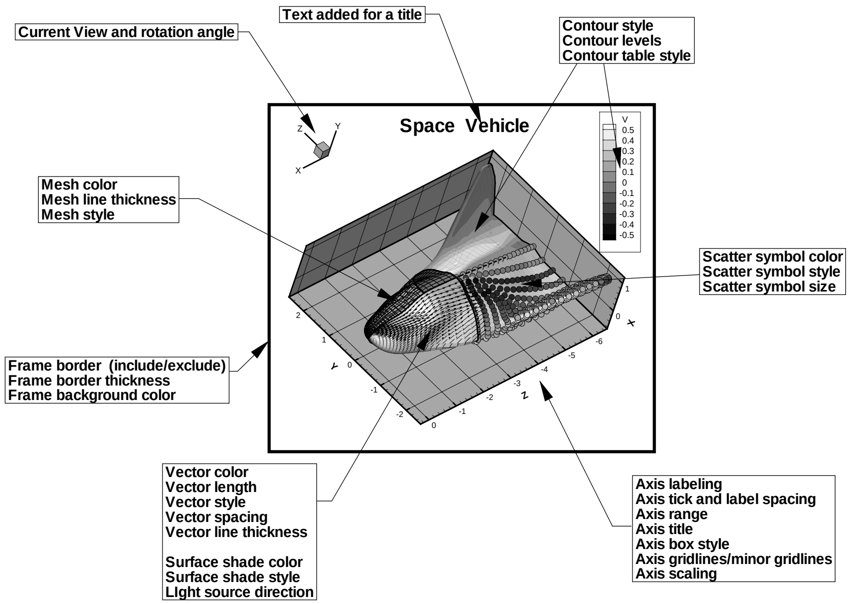



**Figure 6-9.** Some of the items considered part of the frame style.

3. (Options) Click Options. The Paste Style Options dialog appears. Specify which style items to paste, then click Close.
4. When you have finished your file and style selections, click OK.

## 6.3.2. Layout Files

A plot often consists of multiple frames and sometimes even multiple data sets. To capture all the information on the plot, you use layout files, which are a special type of Tecplot macro file (usually with an extension of "`.lay`") that give a complete description of your Tecplot plot, including the names of the data files used to create the plot, the frame layout and data set attachments, axis and plot attributes, the current color map, and so on.

If you want to include the field data with the layout, you can use a layout package file. For more information, see Section 6.3.3, "Layout Package Files."

For complete details on the structure of Tecplot macro files and the commands available in Tecplot layout files, see Chapter 28, "Using Macros," and the *Tecplot Reference Manual*.

Figure 6-10 shows a layout using four frames. The frame in the upper left hand corner is attached to data set 1. The two frames on the right are both attached to data set 2. The frame in the lower left is not attached to any data set.

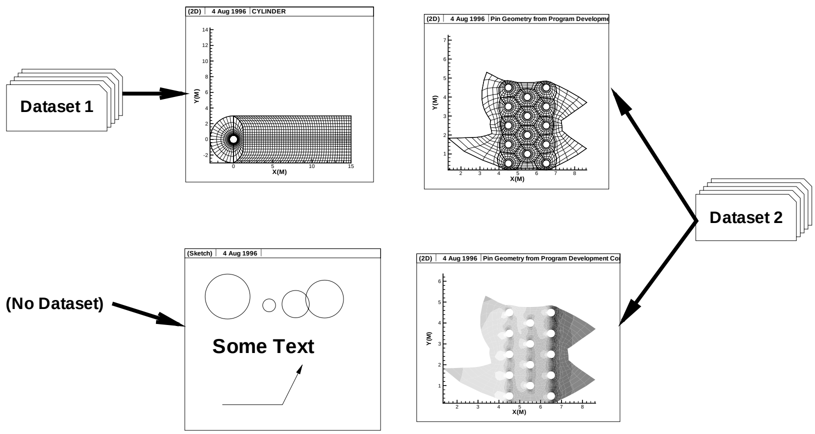



**Figure 6-10.** Layout of four frames using two data sets.

If a frame defined in a layout file requires an attached data set, the data files necessary to build the data set are referenced in the layout file. These data files can be referenced using absolute paths or relative paths. Use relative paths if you intend to move both the layout file and the data files to some other location on disk, or some other platform, at a later date. When using relative paths under Windows, the data files must be on the same drive as the layout file.

Aside from the commands needed to build the individual style of each frame, layout files also include macro commands to set the following:

- Page layout information including, for example, the size and orientation of the paper.
- Some print setup information, including, for example, how colors are to be mapped to monochrome gray scales for monochrome output.
- Color spectrum information, including what basic global color map is installed and what adjustments have been made.

**6.3.2.1. Saving Layout Files.** You save layout files using the Save Layout or Save Layout As options under the File menu. To save your layout, do the following:

1. From the File menu, select Save Layout As. The Save Layout dialog appears.
2. Choose “Linked Data (*.lay)” using the File of type drop-down, as shown in Figure 6-11.

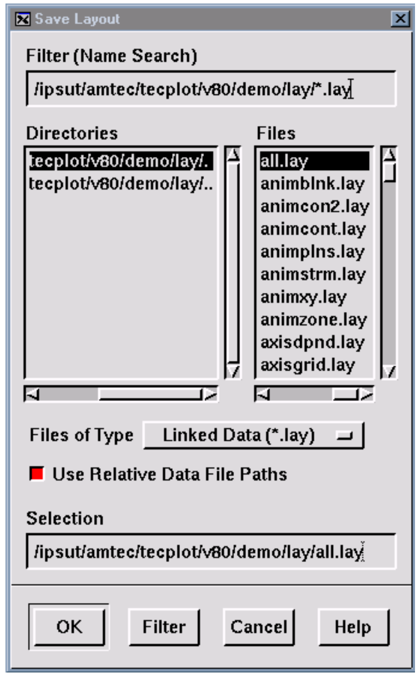



**Figure 6-11.** The Save Layout dialog in Motif, showing the Linked Data (*.lay) option.

3. By default, Tecplot saves the name of the data files used in the layout with their relative file paths. If you want to save your layout using absolute file paths, turn off the Use Relative Path check box.
4. Specify a file name for the layout file. In Windows, you may specify a URL by selecting the URL check box. When selected, you may enter a full URL as the file name, as shown in Figure 6-12. The URL must start with either ftp:// or http://. For example, ftp://ftp.microsoft.com/myplot.plt. Note that the two file types are mutually exclusive. You cannot browse disk files when the URL check box is selected, and you cannot enter a URL if the URL check box is not selected. To open (or save) the URL, click Open URL (or Save URL).
5. If you have changed a data set since you last read it in or wrote it out, Tecplot prompts you for a file name under which to save the changed data. If your layout has multiple data sets, Tecplot prompts you for a file name for each modified data set.

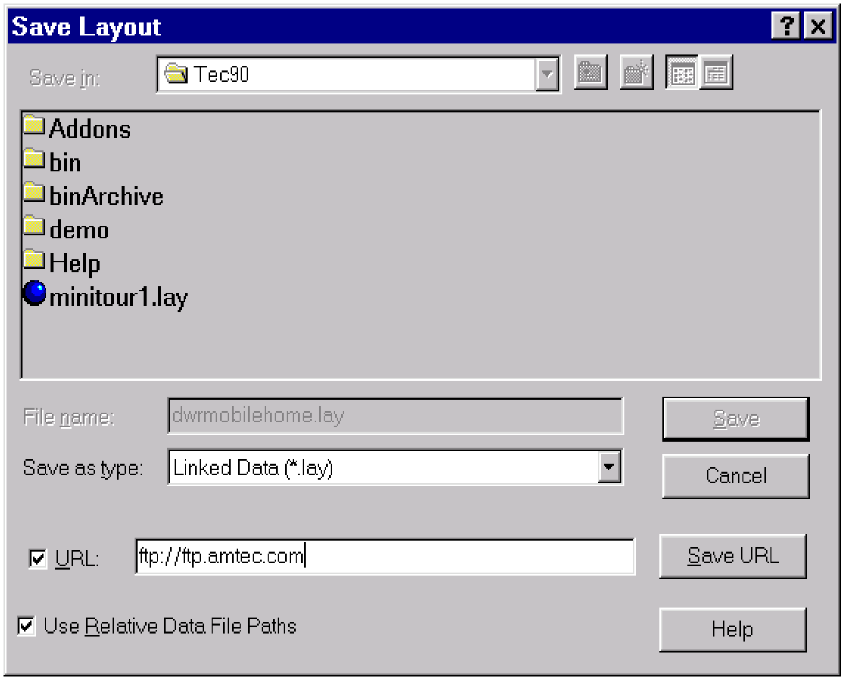



**Figure 6-12.** The Save Layout dialog in Windows, showing the URL check box.

Once you save a layout, the layout file name appears in the header of the Tecplot application window. To save your layout to this file again, simply choose Save Layout from the file menu.

**6.3.2.2. Opening Layout Files.** You open layout files using the Open Layout option under the File menu. To open your layout file:

1. From the File menu, select Open Layout. The Open Layout dialog appears.
2. Specify a file name, then click OK. If you have an unsaved layout, Tecplot asks you if you want to save it before opening a new layout file.
3. (Optional) To combine the layout file with the current layout in Tecplot, click on the Append check box prior to clicking OK.

**6.3.2.3. Opening Layout Files with Different Data Files.** When you open your layout files in Tecplot, you have the option of overriding the data files that are referenced in the layout file. This does not change the layout file, however, you can save a layout file with the new data files.

This section describes the process for overriding a data file from the Open Layout dialog. A second method is described in Appendix A, Section A.5, “Overriding the Data Sets in Layouts by Using "+" on the Command Line.”

To open a layout file with different data files than those specified in the layout file:

1. From the File menu, choose Open Layout. The Open Layout dialog appears.
2. Select the Data Override check box.
3. Specify the layout file name, and click OK.
4. The Override Layout Data dialog appears, as shown in Figure 6-13.
5. One line is listed for each data set in the layout file. Each line contains the data reader name (**TECPLOT** for Tecplot-format data files). If the data set is being loaded by the Tecplot reader, then this line will also list the number of files that make up the data set, and a partial list of file names. If a data loader add-on is used then the instruction used by the loader are listed here. This could be a list of file names just like the Tecplot loader would show.
6. To change the data files or instructions which make up a data set, either double-click on the relevant line or select the line and click Change.
7. Generally, at this point you will be presented with one or more dialogs that allow you to change the list of file names or instructions. For Tecplot-format data files, you will get a file dialog with which to select a new file or files. If the data loader responsible for loading this data set does not have the capability to override the instructions, then an error message will result.
8. Repeat Steps 6 and 7 until the Override Layout Data dialog lists the files which you want to load with your layout file.
9. Click OK to open the layout file with the specified data files (Figure 6-13).

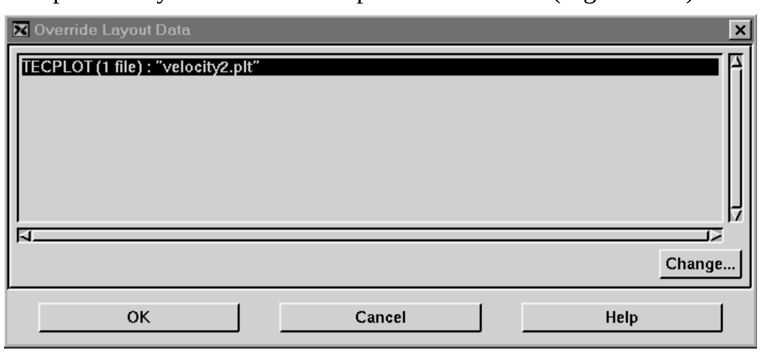



**Figure 6-13.** The Override Layout Data dialog.

### 6.3.3. Layout Package Files

Layout package files are an excellent way of working in collaboration with your colleagues. When sending an image of your data is not enough, they allow you to transmit raw data, along with style information, so your colleagues can view the results in Tecplot. With layout package files the view can be changed, different plot types tested, and so forth.

Layout package files are very useful if you are making large documents containing many images, or other situations when you need to catalog your images. If you save your figures in layout package files, you can quickly view the contents by using the **`lpkview`** utility. This utility allows you to look at thumbnail sketches of each image in a layout packing file without having to load each separately into Tecplot. For more information on **`lpkview`**, see Section 6.3.3.3, "Layout Package Utility."

Layout package files have the same properties as standard layout files. (For more information about layout files, see Section 6.3.2, "Layout Files.") In addition, layout package files also contain all data associated with frames in the layout, and an optional preview image of the Tecplot workspace. To help distinguish layout package files from layout files an extension of **`.lpk`** is used.

**6.3.3.1. Saving Layout Package Files.** You save a layout package file using the Save Layout or Save Layout As options under the File menu. To save your layout package file, do the following:

1. From the File menu, select Save Layout As. The Save Layout dialog appears.
2. Choose "Packaged Data (*.lpk)" from the Files of type drop-down, as shown in Figure 6-14.
3. Choose whether or not you want a preview image included with your layout package file.
4. Specify a file name for the layout package file.

Once you save a layout package file, the layout package file name appears in the header of the Tecplot application window. To save your layout package to this file again, simply choose Save Layout from the file menu.

**6.3.3.2. Opening Layout Package Files.** You open a layout package file using the Open Layout option from the File menu. Note that you may open both standard layout files and layout package files from this dialog.

The Data Override check box is only applicable when the file being opened is a standard layout file. If you select Data Override, and the file you open is a layout package file, you will get a warning dialog and Tecplot will proceed to load in the layout package with the original data.

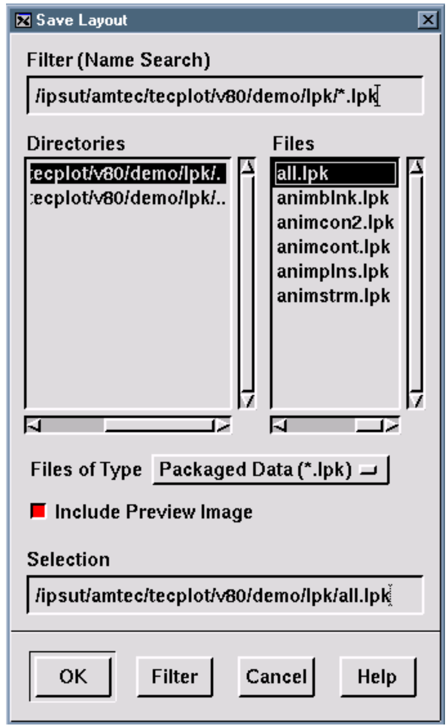



**Figure 6-14.** The Save Layout dialog in Motif, showing the Packaged Data (*.lpk) and Include Preview Image options.

**6.3.3.3. Layout Package Utility.** As a convenience a command line utility, **`lpkview`**, is also provided to catalog, preview, and unpack layout packages. Except for the preview capability all options are consistent between Unix and Windows platforms. The preview image capability varies slightly between the two platforms such that the utility integrates into the target operating system naturally.

Under most Windows operating systems, save for Windows 2000, you can view **`.lpk`** files by:

1. In any explorer window, select the file, then right-click.
2. Select Quick View from the Context menu.

In its simplest form only a layout package file is given to the utility and all default options are assumed. Using default options will cause the utility to unpack the preview image (if present), the layout, and all associated data files into the directory in which the utility was run. For example:

```
lpkview myplot.lpk
```

might unpack the following files in the current directory:

```
myplot.png
myplot.lay
welldata.plt
grid_1.plt
grid_2.plt
```

Names given to the unpacked files were determined by Tecplot when the package was created. Tecplot guarantees that no name conflicts exist within the package by appending unique numbers to non-unique names. However, no attempt is made by **lpkview** to ensure that names are unique with other files located in the directory in which the items are unpacked.

Continuing with the illustration above, if only the layout and associated data files are desired then run the command as follows:

```
lpkview -l -d myplot.lpk
```

Using the **-t** options will output the names of the items within the layout package file without unpacking them. For example:

```
lpkview -t wingperf.lpk
```

might output the following catalog to standard output:

```
wingperf.png
wingperf.lay
run_alpha12beta5.plt
```

For a complete list of lpkview command lines, see Appendix B.2, "LPKView."

## 6.4. Publishing Plots on the Web

Publish enables collaborative research between individuals around the world, within work groups, and among large enterprises. Your results can be used online by saving directly to an HTML file, from which you may read and write data and plot layout files to ftp:// and http:// sites. A Tecplot HTML file could include a reference to a layout package file of your analysis, enabling other Tecplot users browsing your files can review your results directly. The Publish Options dialog and a Tecplot HTML file are shown in Figure 6-15.

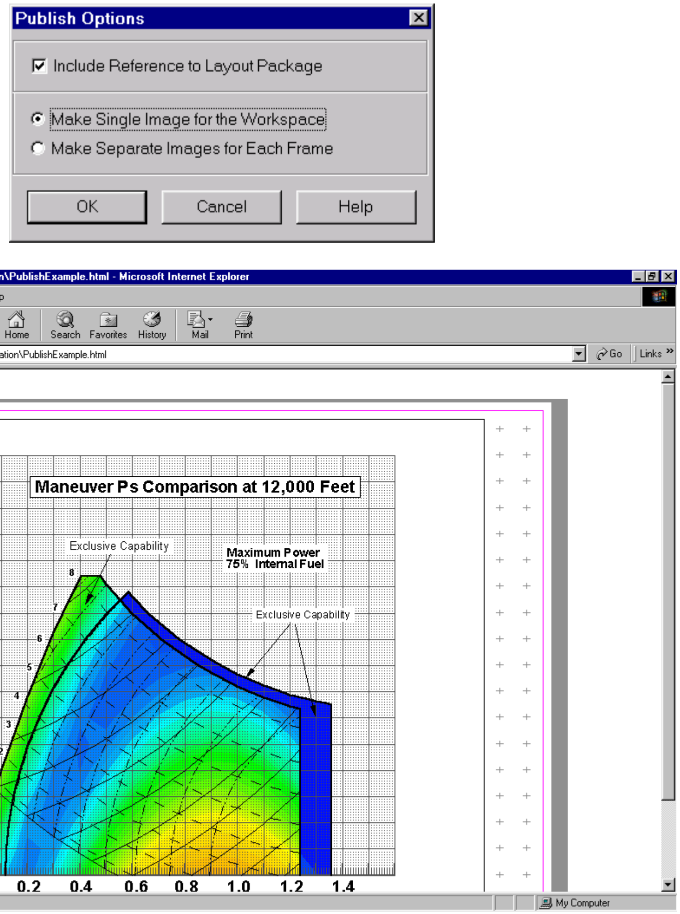



**Figure 6-15.** Tecplot's Publish Options dialog (top) and a Publish HTML file read in a Web browser (bottom). The layout package file may be downloaded and brought into Tecplot by other users.

The Publish feature of Tecplot provides a way in which you can create an HTML file which references the plot images in your Tecplot workspace. Publish can also create a layout package file which includes a link from the HTML file to the layout package file.

To create a Publish file:

1. Choose Publish from the File menu.

2. Decide whether you would like to reference a layout package file. Selecting this option creates a layout package file, along with a reference to that file in the resulting Publish file.
3. Select what images are to be referenced in your Publish file.

   Selecting the Make Single Image for the Workspace check box creates a single image file of your entire Tecplot workspace. A single reference is added to your Publish file for this image file.

   Selecting the Make Separate Images for Each Frame check box creates an image for each frame in your Tecplot workspace. A separate reference for each frame is added to your Publish file.
4. Click OK. You will be presented with a standard file input/output dialog. The name you enter for your file will be used to create the HTML file, create the associated image files, and optionally, the layout package file.

## 6.5. Other Tecplot Files

In addition to the basic ASCII and binary data files, Tecplot uses a number of other files:

- **Equation:** Equation files are used to store equations for data manipulation. They are discussed in their entirety in Chapter 24, “Data Operations.”
- **Macro:** Macro files are used to record and play back Tecplot operations and to set up Tecplot for animation or batch mode. See Chapter 28, “Using Macros,” for more details, and the *Tecplot Reference Manual* for an annotated list of all macro commands.
- **Color map:** A color map file is a Tecplot macro file that saves and restores RGB color values used for contour flooding and multi-coloring. See Chapter 11, “Creating Contour Plots,” for details on creating and modifying Tecplot color map files.
- **Print:** Print files are files created with Tecplot (or Windows) printer drivers. See Chapter 22, “Printing Plots,” for more details.
- **Export:** Export files are graphics files created by Tecplot for import into graphics editing or word processing programs. All the print file formats are available for export, as are several types of bitmaps, Windows Metafile (WMF) Format, and Encapsulated PostScript (EPS) format. For more information on creating bitmap files for export, see Chapter 23, “Exporting Plots.”
- **Curve-coefficient:** These files contain the coefficients for the equations used to draw curves in XY-plots. These are output files only; they cannot be read back into Tecplot. See Chapter 8, “XY-Plots,” for more details.

# CHAPTER 7 Data Loaders: Tecplot's Import Feature

Tecplot allows you to load data in a number of formats with loaders that Amtec has produced using the Add-on Developer's Kit. The Import option on the File menu accesses a scrolled list of data loaders. This chapter tells you how to load data in the following formats:

- **Computational Fluid Dynamics General Notation System (CGNS).**
- **Digital Elevation Map (DEM).**
- **Digital eXchange Format (DXF).**
- **Excel (Windows only).**
- **Fluent Version 5 (`.cas` and `.dat`).**
- **Gridgen.**
- **Hierarchical Data Format (HDF).**
- **Image.**
- **PLOT3D.**
- **Text spreadsheet.**

Available data loaders can be accessed using the Import option on the File menu (Figure 7-1).

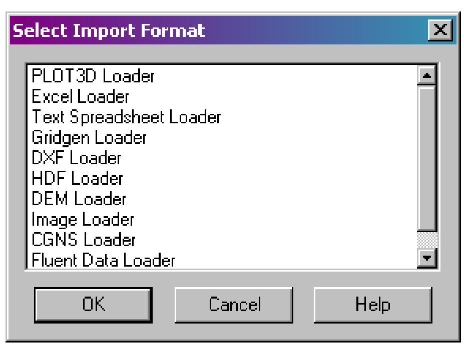



**Figure 7-1.** The Import dialog, accessed via the File menu.

As Amtec writes data loaders they will be posted at our Web site, www.amtec.com. You can also build your own data loaders using the Add-on Developer's Kit.

## 7.1. The CGNS Loader

The CGNS Loader allows you to read CGNS files created with CGNSLib Version 1.2 (Revision 6, June 5, 2000) or earlier into Tecplot. New versions of the CGNS Loader are created to correspond to new releases of the CGNSLib. Check our Web site at www.amtec.com for updates. You are able to choose either all or specific bases, zones, and solutions to be loaded into Tecplot zones. You can also select field variables individually. For structured-grid zones you are allowed to define index ranges to load specific subzone blocks or planes.

Only CGNS bases and zones with valid grids can be read by the CGNS data loader. For unstructured grids Version 2.0 of the CGNS loader supports TRI_3, QUAD_4, TETRA_4, PYRA_5, PENTA_6, HEXA_8, MIXED element types and their combinations on every section. However, the CGNS Data Loader does not support higher-order element types.

Only vertex and cell-center field variable locations are supported. Cell-centered data is averaged to the nodes when the file is read. For cell-centered structured grids arithmetic averaging is used. Rind data is used in the averaging if it is available. For cell-centered unstructured grids either a Laplacian averaging or arithmetic averaging can be selected to average the cell data to the surrounding nodes.

The CGNS Loader dialog is shown in Figure 7-2.

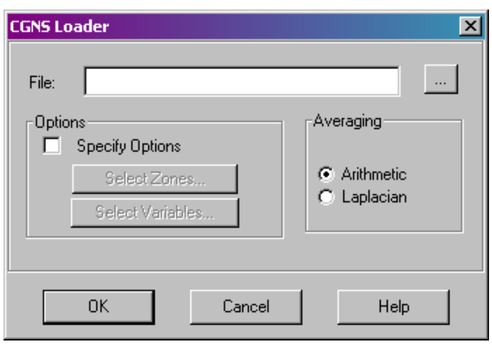



**Figure 7-2.** The CGNS Loader dialog.

The following options are available:

- **File:** Enter the name of the CGNS file to load, including the complete path, or click "**...**" to browse for the file.

- **Specify Options:** This check box is active when a valid CGNS file has been entered or selected. If this option is not selected, CGNS loader will load every base, zone, solution, and variable into Tecplot when you click OK. This option allows you to control the data loaded from your CGNS file, including the ability to load only particular zones, field variables, or partial zones.
- **Select Zones:** Launches the Load CGNS Options: Zones dialog, which allows you to select specific zones and partial zones to load. The dialog is shown in Figure 7-3.
- **Select Variables:** Launches the Load CGNS Options: Variables dialog, which allows you to select specific field variables to load. Grid variables are always loaded automatically. The dialog is shown in Figure 7-5.

### 7.1.1. CGNS Loader Options: Zones Dialog

This dialog is used to specify the zones to load from the data file, and is shown in Figure 7-3.

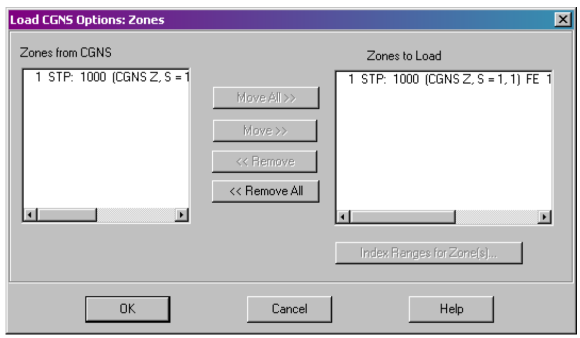



**Figure 7-3.** The CGNS Loader: Zones dialog.

Zones found in the data file are listed under Zones from CGNS. Tecplot zones are not always equivalent to CGNS zones. For example, each solution for a CGNS zone is considered a unique Tecplot zone. The CGNS base, zone, and solution hierarchy orders the zones in this list. The integer that precedes the word “Zone” is the Tecplot zone number that has been assigned to that zone. The integer after the word “Zone” represents the order in which the zone was found in the CGNS data file. The zone description includes the CGNS hierarchy information in parentheses, where “CGNS B, Z, S =” followed by three integers represents the CGNS order for the base, zone, and solution, respectively. If a single base is found, then just “CGNS Z, S =” and two integers are displayed. The description also indicates whether the zone is Ordered (structured) or FE (finite-element or unstructured). The I-, J-, and K-dimensions are provided for ordered zones, and the number of nodes and elements are provided for finite-element zones.

By default all zones are selected for reading and are listed under Zones to Load. This list can be deleted by clicking Remove All. You can also selectively remove zones from the Zones to Load list by highlighting the zone(s) with a click, click-and-drag, Ctrl-click, or Shift-click and then clicking Remove. This will not affect the order of the zones in the Zones from CGNS list. However, the zones that do not appear in the Zones to Load list will not have a Tecplot zone number assigned to them. Similarly, all zones can be added to Zones to Load by clicking Move All, or they can be added selectively by highlighting the zone(s) to be added in the Zones from CGNS list and then clicking Move. Zones in the Zones to Load list will be listed in the order in which they were added to the list regardless of their order in the Zones from CGNS list. This order is the basis for their Tecplot zone number, which is how they will be identified in Tecplot.

To load a partial zone or sub-zone first highlight the zone(s) of interest in the Zones to Load list, then click Index Ranges for Zone(s). This will bring up the Load CGNS: Index Ranges dialog.

## 7.1.2. CGNS Loader Options: Index Ranges Dialog

This dialog is used to specify a subset of the selected ordered/structured zone(s) to be loaded and is shown in Figure 7-4.

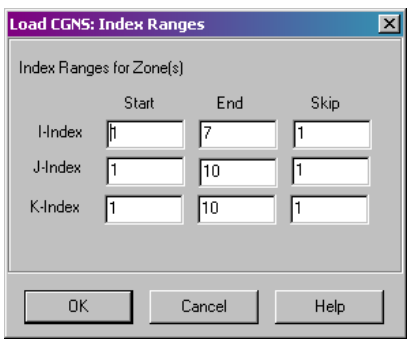



**Figure 7-4.** The CGNS Loader: Index Ranges dialog.

It allows you to define a block, plane, or line of points that will be extracted when the data is loaded. This is accomplished by inputting the ranges for the I, J, and K indices. For each index you must input the Start, End, and Skip values. The Skip value is used to reduce the number of points that will be loaded by skipping points. A Skip value of 3 will load every third point after the Start point. The Start and End points are always loaded. If multiple zones have been selected prior to entering this dialog, the default values for End will be Mx, indicating the maximum value for each zone. You can enter any value for End, but it will only be used in the zones for which it is meaningful. If the value is greater than the maximum index for a zone, then the End for that zone will be replaced by that maximum index.

For multi-dimensional zones the number of points that can specified to load for the I- and J-directions must be greater than one. If the inputs for Start, End, and Skip result in a single point in either direction, an error message will be displayed and you will be forced to change your inputs or cancel the dialog.

### 7.1.3. CGNS Loader Options: Variables Dialog

The Variables dialog contains two lists: Variables from CGNS and Variables to Load, and is shown in Figure 7-5.

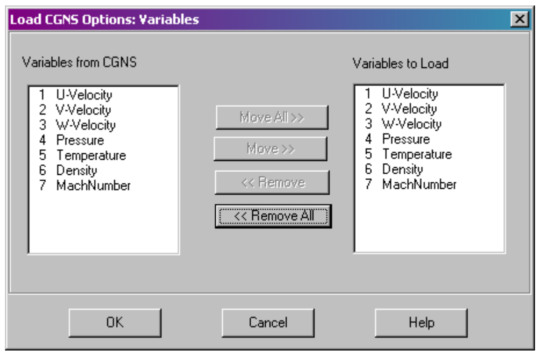



**Figure 7-5.** The CGNS Loader: Variables dialog.

The Variables from CGNS list includes all field variables that were found in the CGNS data file regardless of the zone(s) to which they belong, since CGNS files may contain zones that have different field variables. The Variables to Load list contains the field variables that have been selected to load into Tecplot. Initially, both lists are the same. A Tecplot variable number is assigned to each CGNS field variable that appears in the Variables to Load list. Since Tecplot requires every zone to have the same number of variables, each zone that is loaded into Tecplot will include every variable in the Variables to Load list. This will occur regardless of whether the zone included that field variable in the CGNS file. The variables that were not originally in the zone will be initialized to zero before they are loaded into Tecplot. Since this can unnecessarily increase the size of the data set in Tecplot, you are cautioned to carefully examine the Variables from CGNS list to guarantee that the field variables selected for the Variables to Load list will be consistent with the zones to be loaded.

Remove All lets you delete all variables from the Variables to Load list. Move All lets you include all variables from the Variables from CGNS list. You can also selectively remove variables from the Variables to Load list by highlighting the variable(s) with a click, click-and-drag, Ctrl-click, or Shift-click, and then clicking Remove. The field variables that do not appear in the Variables to Load list will not have a Tecplot variable number assigned to them. Similarly, variables can be added to the Variables to Load by selectively highlighting the variable(s) to be added in the Variables from CGNS list and then clicking Move. Note that vari-

ables in the Variables to Load list will be listed in the order in which they were added to the list regardless of their order in the Variables from CGNS list. This order is the basis for their Tecplot variable number, which is how they will be ordered in Tecplot.

## 7.2. The DEM Loader

The DEM Loader add-on can load Digital Elevation Map files that have the same file format as the U.S. Geological Survey's standard DEM format. These files are generally used by cartographers and geologists to map terrain. The DEM Loader will not accept Spatial Data Transfer Standard (SDTS) formatted data.

DEM files are available on the Web for a number of states within the U.S. For more information, refer to the following references:

- **General:** edcwww.cr.usgs.gov/doc/edchome/ndcdb/ndcdb.html.
- **User's guide:** edcwww.cr.usgs.gov/glis/hyper/guide/l_dgr_dem.

The DEM Loader first launches a multi-file selection dialog. After choosing one or more DEM files to load, you are presented with a simple dialog where you can set the I- and J-skipping. Since DEM files are quite large, you will likely want to set both of these to be 10 or more.

Figure 7-6 shows the DEM Loader dialog.

## 7.3. The DXF Loader

The DXF Loader add-on can import AutoCAD DXF (drawing interchange) files. When importing a file, Tecplot will create an appropriate geometry for each of the following entity types:

- **Text.**
- **Lines.**
- **Arc.**
- **Circle.**
- **Point.**
- **Solid.**
- **3-D faces.**

**Note:** When importing a DXF file, no zones are created. Instead, the geometries representing each entity type are simply added to the frame. Be aware that a typical DXF file can contain several thousand geometries, and these are all included when you save a layout file.

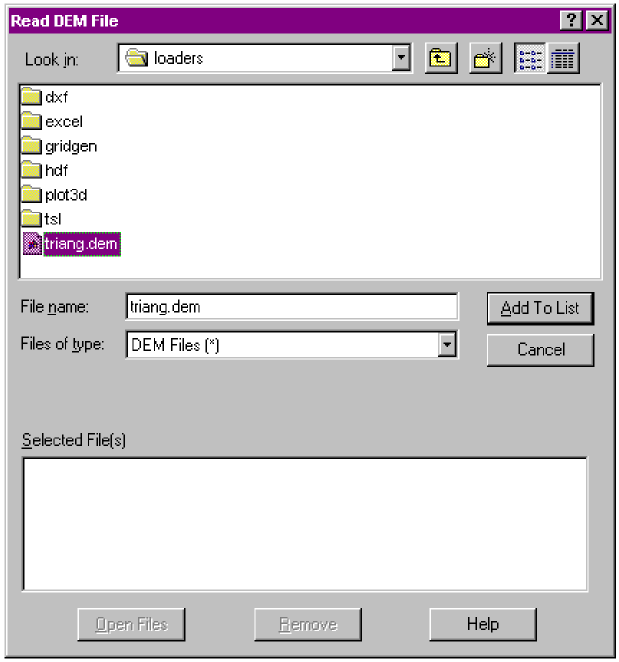



**Figure 7-6.** The Digital Elevation Map (DEM) Loader dialog.

## 7.3.1. The Load DXF File Dialog

The Load DXF File dialog (Figure 7-7) has a variety of features, most of which are self-explanatory.

You can select any of the following:

- **Import:** Select any or all geometries to import -- Text, Lines, Arcs, Circles, Points, Solids, 3D Faces.
- **Font:** Select the font to use for text.
- **Attach Imported Items to Zone:** Specify a zone to which all imported geometries will be attached. Clicking the Select Zone button produces a menu of zone options.
- **Polylines/Import as 2D:** All lines and polylines are stored with three coordinates in DXF files. If you select this option, the loader will add 2-D line geometries for all lines and polylines in the DXF file (the third coordinate will be ignored).
- **Polylines/Import as 3D:** If you select this option, the loader will add 3-D line geometries for all lines and polylines in the DXF file. To view a 3-D DXF file, create or load a 3-D zone, import your DXF file, then choose Fit to Full Size from the View menu.

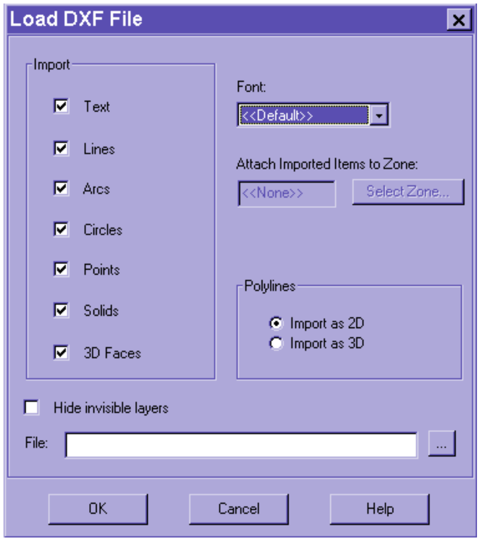



**Figure 7-7.** The Load DXF File dialog.

- **Hide Invisible Layers:** If this option is checked, objects in layers which are "off" in the DXF file will be imported with the background color.

### 7.3.2. Limitations of the DXF Loader

The DXF Loader does not create any field data. Loading a DXF file only adds geometries to your existing frame.

Since most geometries in Tecplot are 2-D, best results will be obtained by loading "flat" DXF files, such as maps.

Binary AutoCAD (`*.dwg`) are not supported in this release.

## 7.4. The Excel Loader

The Excel Loader add-on can read numeric data from Microsoft Excel version 5.0 or higher `.xls` files and import the data into Tecplot (Figure 7-8). The Excel Loader is available only for Windows platforms.

The loader is a point-and-click operation if your spreadsheet is arranged in either of two ways, which we refer to as table format and carpet format. Other formats can also be loaded and

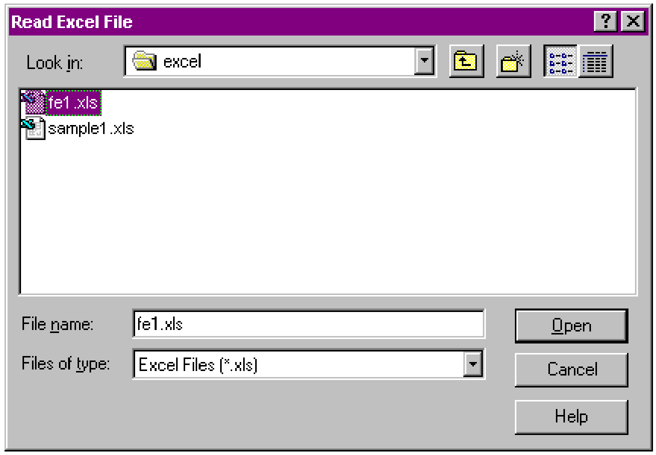



**Figure 7-8.** The initial Excel Loader dialog.

imaged, but they require a little more work, and in some cases more sophistication, on your part.

Once you have chosen an Excel file to load into Tecplot, the Excel Loader leads you through a series of dialogs that let you specify a variety of attributes such as the format of the data in the Excel spreadsheet, the variables to read into Tecplot, and zone information.

## 7.4.1. Spreadsheet Data Formats

The Excel Loader will automatically identify blocks of data in table or carpet format, that is, blocks that satisfy the conditions of these formats. The characteristics of these formats are described in Section 7.4.1.1, “Table Format,” and Section 7.4.1.2, “Carpet Format.” If the loader has identified any carpet or table format blocks, you may select them from a list.

The loader will list blocks of data in standard Excel notation. For example, a block found on worksheet *sheet1*, cells A1-D8, is listed as follows: **`(sheet1 ! A1:D8)`**. It is important to verify that the matrix specified is actually the matrix of data you wish to load.

If you select a user-defined format (or if the loader did not identify any carpet or table blocks), then you will be prompted to enter the names and number of variables, and one or more zones and associated properties. For each zone you will also have to enter the location of the field data in the spreadsheet.

**7.4.1.1. Table Format.** Table format is especially applicable to spreadsheets containing data that will be plotted in XY frame mode—generally, data that represent an independent and one or more dependent variables. Many spreadsheets containing data to be plotted in 2D or 3D

frame modes will also satisfy the conditions of table format. A table formatted data set has the following characteristics:

- The data set is arranged in one or more adjacent columns.
- Each column is the same length and contains numeric data.
- At the top of each column is a variable name (that is, a cell containing the text label of the variable).
- The spreadsheet data set is imported as a single I-ordered zone in **`POINT`** format with *N* variables, where *N* is the number of columns in the table.
- There must be no blank cells within the block of data. An empty cell will prevent the loader from recognizing the block. You can satisfy this condition by filling blank cells with 0.0.
- The block of data must be surrounded by empty cells, text-filled cells, or table boundaries. The loader will not recognize a block of data as being in table format if any cell adjacent to the block is filled with a number.

Figure 7-9 shows an Excel block in table format.

| | A | B | C |
|---|---|---|---|
| 1 | Month | Seattle Rainfall | |
| 2 | 1 | 4.3 | |
| 3 | 2 | 4.5 | |
| 4 | 3 | 4 | |
| 5 | 4 | 4.2 | |
| 6 | 5 | 3.5 | |
| 7 | 6 | 2.1 | |
| 8 | 7 | 2 | |
| 9 | 8 | 1.5 | |
| 10 | 9 | 2.1 | |
| 11 | 10 | 2.5 | |
| 12 | 11 | 3.3 | |
| 13 | 12 | 3.5 | |
| 14 | | | |

**Figure 7-9.** A block of data in table format. Note that for the block to be recognized as such by Tecplot, it must be bounded by spreadsheet edges, text, or empty cells.

**7.4.1.2. Carpet Format.** A spreadsheet to be plotted in 2D or 3D frame mode is likely to be in carpet format. The carpet formatted data set, shown in Figure 7-10, has the following characteristics:

- The spreadsheet data set is imported as an IJ-ordered zone. In Figure 7-10, the spreadsheet is imported as *I=4* and *J=4*. The three variables are X, Y and V. In the spreadsheet cell 2B is index **`1`**, **`1`**, cell 3B is index **`2`**, **`1`**. See section 4.2.2, "IJ-Ordered Data."

- The top row in the block contains the values of the X-variable, the first column of the block contains the values of the Y-variable, and the V-values are the interior data. This format is useful if your data set was generated from a function *f*, such that *f*(X, Y) = V. This may be a simple arithmetic function of X and Y, or may represent measurements of some variable at points on a grid.
- The block is a rectangular arrangement of numeric data in the spreadsheet, with a blank cell in the upper left hand corner.
- There must be no blank cells within the block of data. An empty cell will prevent the loader from recognizing the block. You can satisfy this condition by filling blank cells with **`0.0`**.
- The block of data must be surrounded by empty cells, text-filled cells, or table boundaries. The loader will not recognize a block of data as being in carpet format if any cell adjacent to the block is filled with a number.

| | A | B | C | D | E | F |
|---|---|---|---|---|---|---|
| 1 | | 1 | 2 | 3 | 4 | |
| 2 | 1 | 1 | 2 | 3 | 4 | |
| 3 | 2 | 2 | 4 | 6 | 8 | |
| 4 | 3 | 3 | 6 | 9 | 12 | |
| 5 | 4 | 4 | 8 | 12 | 16 | |
| 6 | | | | | | |

**Figure 7-10.** The carpet table shows values as a simple arithmetic function of X and Y.

**7.4.1.3. Other Formats.** The Other format option gives you a great deal of flexibility in loading data into Tecplot, but also requires you to give the loader more information about the block of data you are loading. A series of dialogs leads you through the process of describing your data, similar to the way you would specify this information in a Tecplot ASCII file. Some of the most relevant attributes of your data set and its format are described in Section 5.1.2.3, “Data Types,” Section 5.2, “Ordered Data,” and Section 5.3, “Finite-Element Data.”

- **Default format:** The Excel Loader offers a semiautomatic option that requires only that you specify the upper left and lower right corners of your data block. Once you’ve specified those corners, it handles the data in the same way that Tecplot handles an unformatted block in an ASCII file. That is, it assumes one zone of I-ordered data in **`POINT`** format.
- **Custom format:** Using the Custom format option, you can specify characteristics of your data set. Custom format has the following features:
  - It allows you to work with spreadsheets containing blank cells or text cells.
  - For XY-, IJ- and IJK-ordered data, you’ll tell the loader the boundaries of the block to load, and how many data points there are within that block (*IMax*, *JMax*, *KMax*).

- For finite-element data, the number of data points is implied by the number of nodes and number of elements.
- Allows you to load blocks of cells that you delimit interactively.
- It is the only option for loading finite-element, IJK-ordered, or zone data from Excel. If a user wants to read in data from an Excel spreadsheet into more than one Tecplot zone the custom format must be used. The default assumes that all data read should be put in a single I-ordered zone.

### 7.4.2. Example: Loading an FEPOINT Excel File in User-Defined Format

The Excel spreadsheet in **`Tec90/examples/loaders/xls/fe1.xls`** (Figure 7-11) contains data in finite-element **`POINT`** format (refer to Section 5.3, "Finite-Element Data," for a discussion of FE-point format). The procedure for loading the data into Tecplot is as follows:

| | A | B | C | D | E | F | G | H | I |
|---|---|---|---|---|---|---|---|---|---|
| 1 | You can load the information on the left as follows: | 0.00E+00 | 0.00E+00 | -9.50E+01 | -1.00E+00 | 1.00E+00 | 0.00E+00 | 8.00E+00 | |
| 2 | | 0.00E+00 | 8.50E+01 | -4.20E+01 | 0.00E+00 | -8.50E+01 | -3.00E+00 | 9.00E+00 | |
| 3 | Variables = "X","Y","Z","C","U","V","W" | 8.10E+01 | 2.60E+01 | -4.20E+01 | 2.00E+00 | -2.20E+01 | 8.00E+01 | 8.00E+00 | |
| 4 | Zone format = FEPOINT | 5.00E+01 | -6.90E+01 | -4.20E+01 | -6.00E+00 | 7.20E+01 | 5.20E+01 | 9.00E+00 | |
| 5 | Element type = TETRAHEDRON | -5.00E+01 | -6.90E+01 | -4.20E+01 | 1.40E+01 | 6.70E+01 | -4.80E+01 | 9.00E+00 | |
| 6 | N = 13 | -8.10E+01 | 2.60E+01 | -4.20E+01 | 2.00E+01 | -3.00E+01 | -8.20E+01 | 9.00E+00 | |
| 7 | E = 20 | 0.00E+00 | 0.00E+00 | 0.00E+00 | 1.00E+00 | -2.00E+00 | -5.00E+00 | 1.00E+01 | |
| 8 | Data Range: B1 - H33 | 5.00E+01 | 6.90E+01 | 4.30E+01 | 1.40E+01 | -6.80E+01 | 4.80E+01 | 1.10E+01 | |
| 9 | | 8.10E+01 | -2.60E+01 | 4.30E+01 | 2.00E+01 | 3.10E+01 | 8.20E+01 | 1.10E+01 | |
| 10 | | 0.00E+00 | -8.50E+01 | 4.30E+01 | 0.00E+00 | 8.40E+01 | 3.00E+00 | 1.00E+01 | |
| 11 | | -8.10E+01 | -2.60E+01 | 4.30E+01 | 2.00E+00 | 2.10E+01 | -8.00E+01 | 1.10E+01 | |
| 12 | | -5.00E+01 | 6.90E+01 | 4.30E+01 | -6.00E+00 | -7.10E+01 | -5.10E+01 | 1.10E+01 | |
| 13 | | 0.00E+00 | 0.00E+00 | 9.60E+01 | 1.00E+00 | 0.00E+00 | -1.00E+00 | 1.20E+01 | |
| 14 | | 1 | 2 | 3 | 7 | | | | |
| 15 | | 1 | 3 | 4 | 7 | | | | |
| 16 | | 1 | 4 | 5 | 7 | | | | |
| 17 | | 1 | 5 | 6 | 7 | | | | |
| 18 | | 1 | 6 | 2 | 7 | | | | |
| 19 | | 2 | 8 | 3 | 7 | | | | |
| 20 | | 3 | 9 | 4 | 7 | | | | |
| 21 | | 4 | 10 | 5 | 7 | | | | |
| 22 | | 5 | 11 | 6 | 7 | | | | |
| 23 | | 6 | 12 | 2 | 7 | | | | |
| 24 | | 12 | 2 | 8 | 7 | | | | |
| 25 | | 8 | 3 | 9 | 7 | | | | |
| 26 | | 9 | 4 | 10 | 7 | | | | |
| 27 | | 10 | 5 | 11 | 7 | | | | |
| 28 | | 11 | 6 | 12 | 7 | | | | |
| 29 | | 12 | 8 | 13 | 7 | | | | |
| 30 | | 8 | 9 | 13 | 7 | | | | |
| 31 | | 9 | 10 | 13 | 7 | | | | |
| 32 | | 10 | 11 | 13 | 7 | | | | |
| 33 | | 11 | 12 | 13 | 7 | | | | |
| 34 | | | | | | | | | |

**Figure 7-11.** Excel file **`fe1.xls`**, used in the example in Section 7.4.2.

1. Select the Import option from the File menu.
2. Choose Excel.
3. In the Read Excel File, specify a path and a file, and click OK.
4. In the Import Excel File—Step 1 dialog, you are restricted to Other format, because **`fe1.xls`** does not satisfy the conditions of table or carpet format. Click Next>.

5. In the Step 2 of 4 dialog, add seven variables of type Double, and a title if you wish. Click Next>.
6. In the Step 3 of 4 dialog, click Add.
7. From the Add menu select the Edit Zone option and specify that:
   - You choose the block of data that extends from B1 to H33.
   - The format of the data file is **`FEPOINT`**.
   - The data set contains 13 nodes.
   - Those nodes are connected into 20 elements.
   - The element type is **`TETRAHEDRON`**.
8. Click OK. The Step 3 of 4 dialog now displays the zone you have described, with a + button that you can press to display your parameters.
9. Click Next>.
10. The Import dialog displays some of your choices. Confirm them and click Finish.
11. The initial plot is in 2D frame mode, which you can convert to 3D mode for a full view of the finite-element volume (Figure 7-12).

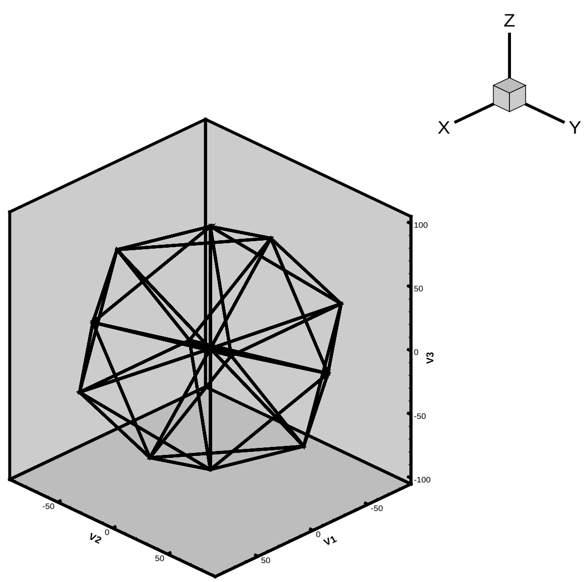



**Figure 7-12.** Excel spreadsheet **`fe1.xls`**, plotted in 3D frame mode.

### 7.4.3. Restrictions of the Excel Loader

Recall that a block of data is a rectangular group of numbers in the spreadsheet. The loader places the following restrictions on blocks:

- Carpet and table format, which the loader detects and loads automatically, are fairly narrowly defined. All other formats must be loaded on the user-defined pathway.
- Numeric cells within each block should contain only numbers or numeric characters such as **+**, **-**, and so forth. For example, a cell which contains *X=34* is interpreted by the loader as text, since it begins with text.
- Cells containing formulas (therefore displaying calculated values) will be skipped by the loader. You can convert the formulas to values within Excel.
- The spreadsheet file must have been written by Excel Version 5.0 or higher. Earlier versions of Excel are not supported.

## 7.5. The Fluent Loader

The Fluent Data Loader allows you to read Fluent Version 5 case (**.cas**) and data (**.dat**) files into Tecplot. To load files from earlier versions of Fluent, you must first import them into Fluent 5, then save them as Fluent 5 files.

Fluent stores solution data at cell centers (face centers for boundary zones). Since Tecplot requires all data to be at the nodes, the Fluent Data Loader averages the cell or face center data to the surrounding nodes using arithmetic averaging. Values at hanging nodes (nodes in the center of a cell face or edge) are currently calculated only from those cells of which the node is a corner. Hanging nodes can lead to discontinuous contours due in part to this one-sided averaging. The Fluent Data Loader dialog is shown in Figure 7-13.

The following options are available:

- **Load Grid and Solution Data:** Loads both a case and a data file. The grid comes from the case file, and the solution comes from the data file. Specify the case file and other grid options in the Grid Options section of the dialog. All variables are read from the data file and added to the Tecplot data set. If a particular variable is present in the data file for only one zone, it is set to zero in Tecplot for all other zones.
- **Load Grid Only:** Loads the grid from a case file. Specify the case file and other grid options in the Grid Options section of the dialog.
- **Load Residuals Only:** Loads the residual data (convergence history) from a data file. The residuals are not scaled or normalized.
- **Grid Options:** The Grid Options portion of the dialog pertains to the Fluent case (**.cas**) file. It contains the following options.

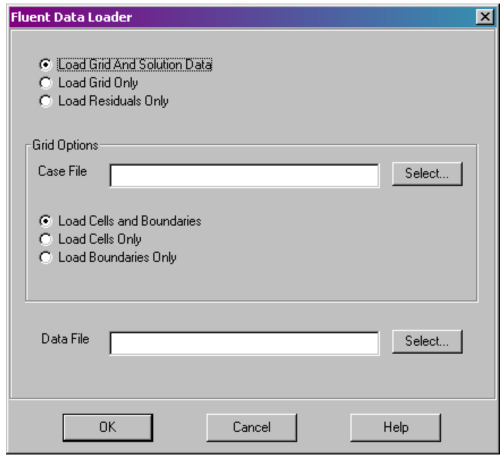



**Figure 7-13.** The Fluent Data Loader dialog.

- **Case File:** Type the name of the case file you wish to load, or click Select, then select the name of the file from the resulting dialog.
- **Load Cells and Boundaries:** Loads the cell (solution) and boundary zones from the case file. Each fluid or solid cell zone and each boundary zone will be displayed as a separate zone in Tecplot.
- **Load Cells Only:** Loads only the cell (solution) zones. Each zone will be displayed as a separate zone in Tecplot.
- **Load Boundaries Only:** Loads only the boundary zones. Each zone will be displayed as a separate zone in Tecplot.
- **Data File:** The data (.dat) file contains the solution and the residual (convergence history) data. Type the name of the data file, or click Select, then select the name of the file from the resulting dialog.

## 7.6. The Gridgen Loader

The Gridgen Loader add-on accepts output from Pointwise, Inc.'s Gridgen Version 11. (Amtec has not tested previous versions.) The Gridgen Loader can import the following types of Gridgen files into Tecplot:

- Database Network (**`*.net`**)—one IJ-ordered zone is created for each network in the file.
- Volume Grid (**`*.dat`**)—one IJK-ordered zone is created for each block of data in the file.

More information on Gridgen Volume Grid and Database Network files can be found in the *Gridgen User's Manual*.

The files can be in any of the following formats, which are automatically detected:

- **ASCII.**
- **Binary formatted.**
- **Binary unformatted.**
- **Single or double precision.**

The data set is given a default title of “Imported Gridgen Data,” which you may change by selecting the Data Set Info option from the Data menu.

Variables names default to “**X**,” “**Y**,” and “**Z**.” These can be changed within Tecplot after the data set is loaded.

The Gridgen Loader leads you through several screens, each of which allows you to specify one or more attributes of the input files (Figure 7-14).

### 7.6.1. Loading Gridgen Data Using Tecplot

The Gridgen Loader dialog asks for the following information:

- **File Type:** Select the type of file you wish to import.
- **I-Skip, J-Skip, K-Skip:** Select the I-, J-, and K-Skip values. A value of 1 will read every data point, 2 will read every other data point, and so on.

After you have selected the file type and skip values, click OK and you will be prompted for one or more files to load. Select one or more files and click OK to load the files.

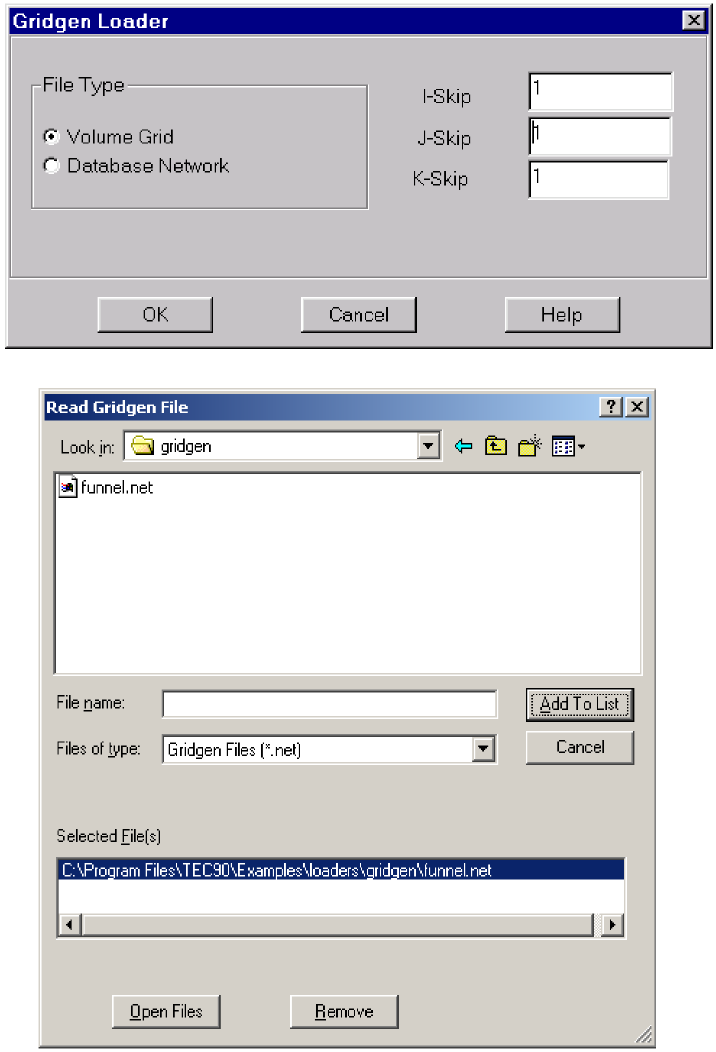



**Figure 7-14.** The Gridgen Loader dialogs.

# 7.7. The HDF Loader

The Tecplot HDF Loader add-on can load 1-D, 2-D, and 3-D Scientific Data Sets (SDS) from HDF files. Its dialog is shown in Figure 7-15.

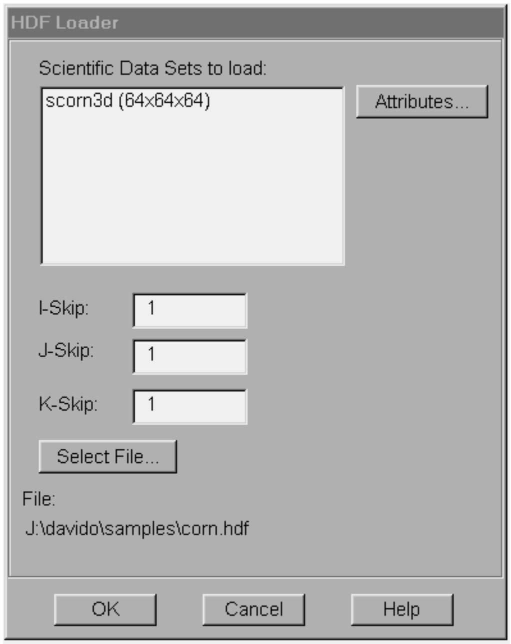



**Figure 7-15.** The HDF Loader dialog.

A data set from an HDF file is imported as follows:

1. The file is scanned and a list of all SDS in the file is created.
2. You select one or more SDS to import. Each SDS that you select must have the same dimension.
3. A rectangular I-, IJ-, or IJK-ordered zone (for 1-, 2-, or 3-D data, respectively) is created for each SDS that you select to load.
4. The data is imported.

The HDF Loader dialog asks the following information:

- **Scientific Data Sets to load:** Select one or more SDS's to load. Each SDS that you select must have the same rank (dimension).
- **I-Skip:** Select the I-Skip value. A skip value of 1 loads every data point, a skip value of two loads every second data point, and so on.

- **J-Skip:** Select the J-Skip value.
- **K-Skip:** Select the K-Skip value.
- **Select File:** Select an HDF file.
- **Attributes:** Displays attributes of each SDS found, such as number type, rank, label, and so on.

**Note:** The HDF Loader uses the public-domain HDF API code library from the National Center for Supercomputing Applications (NCSA), University of Illinois, Urbana-Champaign.

### 7.7.1. HDF Loader Limitations

The HDF Loader can import only Scientific Data Sets from HDF files, and these are imported in a manner similar to NCSA's own HDF viewer. The way in which the data file is interpreted cannot be altered in this release of the loader. However, it is possible to write a Tecplot Version 9.0 add-on (using the NCSA code library) which loads HDF data in a manner more suited to your particular use of the HDF format. See the *ADK User's Manual* for more information on writing add-ons.

## 7.8. The Image Loader

The Image Loader add-on allows you to import a **.bmp** file as a group of Tecplot geometries. When importing a **.bmp** file, Tecplot will create one or more rectangular geometries for each line of the image. The Image Loader dialog is shown in Figure 7-16.

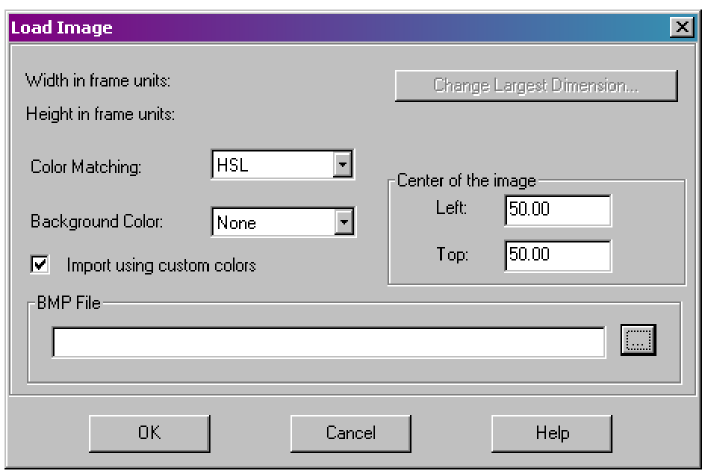



**Figure 7-16.** The Image Loader dialog.

The following options are available on the Image Loader dialog:

- **Width in Frame Units:** When the image is imported, it will be scaled to this with. You may change this value by selecting Change Largest Dimension.
- **Height in Frame Units:** When the image is imported, it will be scaled to this height. Width and height are calculated so that the original aspect ratio of the image is preserved. Thus you may only change the largest dimension. The other dimension will be calculated based on this setting.
- **Change Largest Dimension:** Click to set the imported width or height in frame units. Only the largest dimension is changed, the other dimension will be calculated for you.
- **Background Color:** If you select a background color, any pixels in the original image which match this color will be ignored during the import. This will both reduce the total number of geometries created and allow certain parts of images to be transparent.
- **Import Using Custom Colors:** If you select this option, Image Loader will try to match the original image colors to Tecplot's custom colors, in addition to Tecplot's basic colors. This will generally result in better imported images.
- **Position Inside the Frame:** You may set the initial position of the center of the imported image inside the Tecplot frame.
- **Color Matching:** Image Loader can match colors using two different algorithms. Select HSL to use the closest distance in HSL (Hue, Saturation, Luminance) color space. Select RGB to use the closest distance in the RGB color space.
- **BMP File:** Select the file to import here.

### 7.8.1. Limitations of the Image Loader

- The image loader does not create any field data, only rectangular geometries. Large images will result in a large number of geometries.
- Colors in the original image are matched as best as possible to the eight basic and eight custom Tecplot colors. However, with only 16 colors to choose from, some images will not look as good as the original when imported. Smaller images which use only the basic colors will work best.
- BMP files must be eight- or 24-bit and cannot be compressed. You may have to re-save the **`.bmp`** image as an eight- or 24-bit uncompressed **`.bmp`** before attempting to import it.

## 7.9. The PLOT3D Data Loader

The PLOT3D Loader add-on can import data files formatted for the PLOT3D program developed by Pieter Buning at the NASA Ames Research Center. Some extensions such as unstructured data that are now available in FAST, the successor to PLOT3D, are also supported.

Figure 7-17 shows the PLOT3D Data Loader dialog. A grid file must be selected and assigned to the text field at the top of the dialog. Click "..." to browse for a file name.

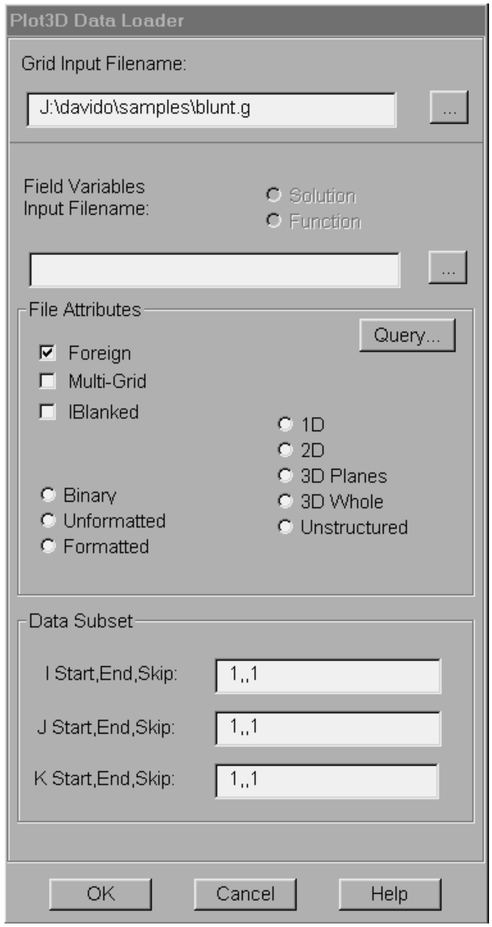



**Figure 7-17.** The PLOT3D Data Loader dialog.

A solution or function file may also accompany the grid file. The solution or function file is specified in the second text field. You must first select a grid file before selecting a solution or function file. The loader cannot detect whether a given file is a solution or function file, so you must specify the type using the solution or function toggles.

### 7.9.1. PLOT3D File Attributes

The PLOT3D file attributes are divided into three groups: option buttons that define the data structure, option buttons that set the file format, and a set of miscellaneous toggles referencing data organization.

### 7.9.2. Setting the Data Structure Attribute

On the right side of the data loader dialog is an area for selecting the data structure. PLOT3D data can be 1-, 2-, or 3-D-planes, 3-D whole, or Unstructured. You may only select one of these options; 3-D whole is the most common.

### 7.9.3. Setting the File Format Attribute

The file format is set using one of the option buttons in the lower left of the File Attributes section. Formatted files are ASCII, and historically have used the **`.fmt`** extension, but are not restricted to doing so. Unformatted files are binary files that contain FORTRAN record markers and historically use the **`.dat`** extension. Binary files are just that, binary, however unlike the unformatted files, contain no record markers and historically use the **`.bin`** extension.

### 7.9.4. Setting Miscellaneous Attributes

In the upper left corner of the File Attributes section are three toggles that set miscellaneous file attributes.

The Foreign toggle must be set if the data was written out on a platform whose binary data byte ordering is foreign to the platform you are running Tecplot on. This typically involves non-Intel machines versus Intel machines. If your files are generated on the same machine on which you are running Tecplot, then you most likely will not want to turn this toggle on.

The Multi-Grid toggle must be set if the data is multi-grid. Each grid will be turned into a separate zone within Tecplot.

The IBlanked toggle must be set if the data contains the extra IBlanking value. A separate variable will be created in Tecplot with the IBlanking value.

### 7.9.5. Determining the File Attributes

Most often you will know the attributes of your data files ahead of time. In the case that you do not know the attributes, a Query function is provided to help you make some educated guesses. Clicking Query calls up a file selection dialog. Select the file you would like to query, and an

informational dialog will appear with some observations about the file. This information may then aid you in filling out the rest of the dialog.

### 7.9.6. Reading In a Subset of the Data

The bottom section of the PLOT3D Data Loader dialog is used to set index skipping. This allows you to load a subset of the data. The index skipping will be applied to all grids in a multi-grid file. You are not allowed to set index skipping for unstructured data.

## 7.10. The Text Spreadsheet Loader

The Text Spreadsheet Loader add-on is both an example of how to write a loader add-on and a utility which lets you import simple data from ASCII files. The complete source code for the Text Spreadsheet Loader is included in the ADK Examples directory.

### 7.10.1. Data File Format

The Text Spreadsheet Loader can read ASCII files of the following format (blank lines are ignored):

```
Variable 1, Variable 2, ..., Variable N
datapoint1,datapoint2, ..., datapoint N
.
.
.
datapoint1,datapoint2, ..., datapointN
```

Here is an example of a valid ASCII spreadsheet file:

```
Month, Rainfall
1, 15.0
2, 21.0
3, 21.0
4, 32.0
5, 10.3
6, 5.1
7, 2.3
8, 0.2
9, 1.4
10, 8.3
11, 12.2
12, 15.4
```

### 7.10.2. Text Spreadsheet Loader Limitations

All of the variable names must be on the first line.

# CHAPTER 8 XY-Plots

This chapter discusses XY-plots in detail. In Tecplot, you may create XY-line plots, symbol plots, and bar charts. You may also include error bars, either by themselves or in combination with any of the other map layers.

An XY-plot is simply a graph of one or more series of independent and dependent (X, Y) data points. Each series of data points is referred to as an XY-mapping or mapping, the Tecplot dialogs often condense this as maps. Each XY-mapping associates one variable with an X-axis and one variable with a Y-axis. For example, you may create an XY-plot of an airplane's speed versus its altitude, with Speed on the vertical Y-axis and Altitude on the horizontal X-axis. To create this plot, you need only one XY-mapping, which associates Altitude with one of Tecplot's five X-axes and Speed with one of Tecplot's five Y-axes, as shown below in Figure 8-1.

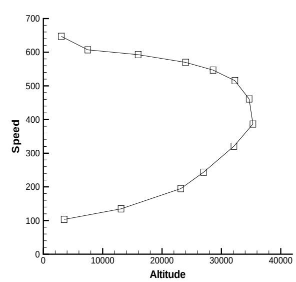



**Figure 8-1.** XY-plot with one XY-mapping.

To create an XY-plot such as Figure 8-1, the procedure to follow is:

1. From the Data menu, select Create Zone.
2. From the Create Zone drop-down, select Enter XY-Values. The dialog titled Enter XY-Values to Create a Zone appears.

3. In the text box labeled Enter XY Values, enter X- and Y-value pairs, one per line; first X, then one or more spaces, then Y.
4. If you would like to specify a data type for the data (integer, float, double, byte, bit), select the desired data type from the drop-down labeled Destination Data Type.
5. After entering all of the XY-values, click Create, then Close.

You might also construct an I-ordered zone in a data file. For example, consider the data file **`simpxy.dat`** (stored in the **`examples/dat`** directory in your Tecplot home directory). This file lists the values of temperature measured at 20 locations unequally spaced along a wire. We would like to plot temperature versus distance along the wire. We can assign an index (*i*) to identify each row of data; every row represents a single data point. Then each data point can be identified uniquely by its I-index. Data point number 1 is addressed as *I=1*, data point number 2 is addressed as *I=2*, and so on up to *I=20*. Data which can be addressed in this way is called I-ordered data.

To plot this data, just read the data file into Tecplot. The default frame mode is XY because this data set is I-ordered.

By default, Tecplot plots the first variable (**`V1`**) on the X-axis versus the second variable (**`V2`**) on the Y-axis. You may use the Define XY-Mappings dialog to reassign variables to the axes.

Tecplot initially sets the ranges on the X- and Y-axes so that you can see all of your data points, (grid mode) text, and (2D grid mode) geometries. This view can be changed using the Edit Axis dialog and the View menu. The standard default axis mode for XY-plots is independent, meaning that the scales on the X- and Y-axes are unrelated. This can be changed using the Edit Axis dialog.

When you initially create an XY-plot, Tecplot assigns colors, symbol types, and line patterns to each line (that is, a series of data points). These and other XY-plot attributes can be changed using the seven linked dialogs known collectively as the XY Plot Attributes dialogs. These include the Define XY-Mappings dialog, along with the Line Attributes, Curve-Fit Attributes, Symbol Attributes, Bar Chart Attributes, Error Bar Attributes, and Index Attributes dialogs.

The initial behavior of XY-plots can be changed in the configuration file to create different default settings for many of the XY-plot options.

## 8.1. XY-Plot Data

XY-plots are usually created from one-dimensional data in the I-ordered structure. Tecplot also allows you to create XY-plots from two- or three-dimensional data in the IJ- or IJK-ordered structure, or finite-element data. Finite-element data sets are treated as I-ordered; the connectivity list is ignored. IJ-ordered data sets are just a family of J sets of I-ordered data; I-ordered data can be thought of as IJ-ordered data with *J=1*. IJK-ordered data sets are just K-planes of J-

families of lines; I-ordered data can be thought of as IJK-ordered data with $J=K=1$. You can make XY-plots of IJ- or IJK-ordered data simply by loading your files in XY frame mode, or clicking XY from 2D or 3D frame mode.

Use the Index Attributes dialog to select different ranges and skip intervals for the I-, J-, and K-indices. See Section 8.9, "Selecting I-, J-, and K-Indices," later in this chapter.

The data used for XY-plots must have at least two variables defined at each data point (one for the X-axis and one for the Y-axis). The same number of variables must be defined at each data point. You may have up to 32,700 variables defined at each data point. For example, you could be recording the pressure at a set of 500 pressure probes every minute for two hours (120 time samples). The data could be organized into an array of numbers with 120 rows and 501 columns. The first column could be the time and columns 2 through 501 the pressure measurements. Each row represents a data point with 501 variables. In Tecplot, you can plot all or some of the pressure probes versus time on one plot. Since you can select any variable as the variable on the X-axis, the pressure at probe number 59 (on the Y-axis) could be plotted versus the pressure at probe number 5 (on the X-axis).

## 8.2. Creating XY-Mappings

All XY-plots in Tecplot are composed of the graphs of one or more XY-pairs. The XY-pairs and their dependency relations are defined in Tecplot as XY-mappings. XY-mappings can be displayed as a combination of one or more of three basic plot styles:

- **Lines**: Tecplot draws line segments connecting all of the data points in order, or a curve that represents a fit of some mathematical function to the data.
- **Symbols**: Each data point is represented by a solid or outline symbol, such as circles, triangles, or squares.
- **Bars**: Each data point is represented by a vertical or horizontal bar, according to whether the dependent variable of the XY-pair is the Y-variable or the X-variable.

Each XY-mapping consists of a pair of variables, one assigned to one of five X-axes and the other assigned to one of five Y-axes. XY-mappings are defined for each frame; the same data set can have a different set of XY-mappings in each frame it is attached to.

You define XY-mappings using the Define XY-Mappings dialog, which you access from the XY menu. (The XY menu is active only if the frame mode is set to XY.)

To define a new XY-mapping:

1. From the XY menu, select Define XY-Mappings. The Define XY-Mappings dialog appears. Figure 8-2 shows the demo file **`rain.plt`**.
2. Click Create Map. The Create XY-Mappings dialog appears, as shown in Figure 8-3.

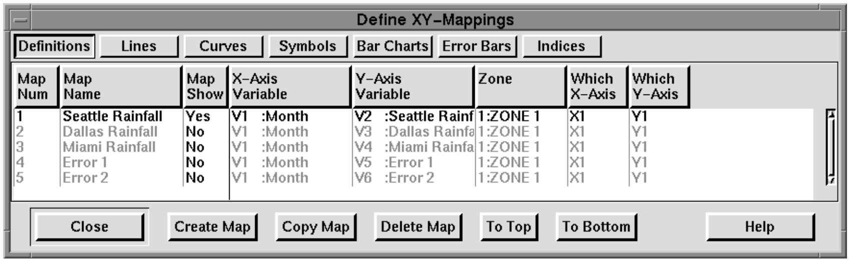



**Figure 8-2.** The Define XY-Mappings dialog.

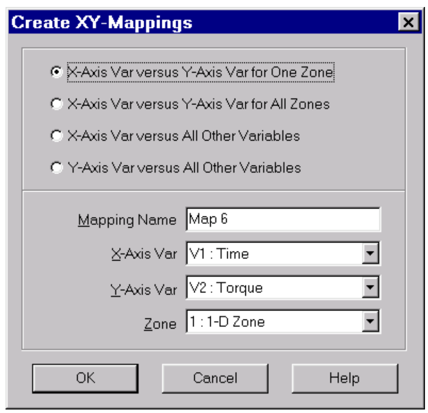



**Figure 8-3.** The Create XY-Mappings dialog.

3. Choose what sort of XY-mapping (or mappings) to add. You have the following options:
   - **X-Axis Var versus Y-Axis Var for One Zone:** (Default) Click this option if you want to add a single XY-mapping with one X- and one Y-variable defined for one zone. If you select this option, continue with Step 4.
   - **X-Axis Var versus Y-Axis Var for All Zones:** Click this option if you want to add one XY-mapping for each zone. You choose an X- and a Y-variable; Tecplot creates an XY-mapping with those variables for each of the current data set’s zones. If you select this option, continue with Step 5.

- **X-Axis Var versus All Other Variables:** Click this option if you want to create a new set of mappings using one variable as the X-variable and each of the other variables in the data set as Y-variables. When you choose this option, you must also specify the zone for which this set of mappings is defined. If you select this option, continue with Step 5.
- **Y-Axis Var versus All Other Variables:** Click this option if you want to create a new set of mappings using one variable as the Y-variable and each of the other variables in the data set as X-variables. When you choose this option, you must also specify the zone for which this set of mappings is defined. If you select this option, continue with Step 6.

4. (Optional) Enter a name for the mapping in the field labeled Mapping Name. The default mapping name is “Map *n*,” where *n* is the number of the mapping that is to be created.
5. Choose an X-axis variable by selecting a variable from the drop-down labeled X-Axis Var. The default is V1.
6. Choose a Y-axis variable by selecting a variable from the drop-down labeled Y-Axis Var. The default is V2.
7. Choose a zone by selecting a zone from the drop-down labeled Zone. The default is the first zone.

When you first read an ordered data set, Tecplot automatically defines some XY-mappings. If your data set has more than two variables, Tecplot creates XY-mappings that associate the first variable with each of the other variables for the first zone only. If your data set has only two variables, Tecplot creates XY-mappings which associate the first variable with second variable for each zone. By default, each of these mappings assigns the first variable to the first X-axis, X1, and the other variable to the first Y-axis, Y1. Tecplot automatically activates the first mapping.

## 8.3. Editing XY-Mappings

To edit an existing XY-mapping, use the Define XY-Mappings dialog. To call up this dialog, choose Define XY-Mappings from the XY menu, or double-click on an existing XY-plot. From this dialog, you can perform the following tasks:

- Modify the names of XY-mappings.
- Activate and deactivate XY-mappings.
- Assign X- and Y-axis variables to selected XY-mappings.
- Assign zones to selected XY-mappings.
- Assign particular X- and Y-axes to selected XY-mappings.

Each of these tasks is discussed in detail in the following subsections.

### 8.3.1. Modifying XY-Mapping Names

Tecplot automatically assigns each XY-mapping a name. The nature of the name varies with the type of data used to create the mapping. If your data has only one dependent variable, the default is to use the zone name for the mapping. If your data has multiple dependent variables, then the default is to use the dependent variable name for the mapping. You can modify any mapping's name using the Enter XY-Mapping Name dialog. This dialog is accessible from all the XY Plot Attributes dialogs by selecting Edit Name from the Map Name drop-down.

To modify an XY-mapping name:

1. From the Define XY-Mappings dialog, select the mapping or mappings for which you want to change the name.
2. Click on the Map Name column header.
3. Click Edit Name to call up the Enter XY-Mapping Name dialog, as shown in Figure 8-4.
4. Enter a new name for the selected mapping or mappings, or construct a new name from text you enter and/or one or more of the predefined inserts:
   - **Zone Name:** Adds the string "**`&ZN&`**" to the Map Name field, which is then replaced with the actual name of the zone assigned to that mapping.
   - **X-Axis Num:** Adds the string "**`&X#&`**" to the Map Name field, which is then replaced with the actual number of the X-axis assigned to that mapping.
   - **Y-Axis Num:** Adds the string "**`&Y#&`**" to the Map Name field, which is then replaced with the actual number of the Y-axis assigned to that mapping.
   - **Independent Var:** Adds the string "**`&IV&`**" to the Map Name field, which is then replaced with the actual name of the independent variable assigned to that mapping.
   - **Dependent Var:** Adds the string "**`&DV&`**" to the Map Name field, which is then replaced with the actual name of the dependent variable assigned to that mapping.

   To add an insert, simply click on its button, or type the associated string directly into the Map Name field.
5. When the Map Name field is as desired, click OK to make the change, or Cancel to abandon the changes.

For example, Figure 8-4 shows the Enter XY-Mapping Name dialog assigning an XY-mapping name consisting of the X-Axis Number, a dash (two hyphens), and the dependent variable name.

### 8.3.2. Activating and Deactivating XY-Mappings

Your XY-plot can show any or all XY-mappings defined for the current frame. You can activate and deactivate mappings from any of the Plot Attributes dialogs using the Map Show drop-down.

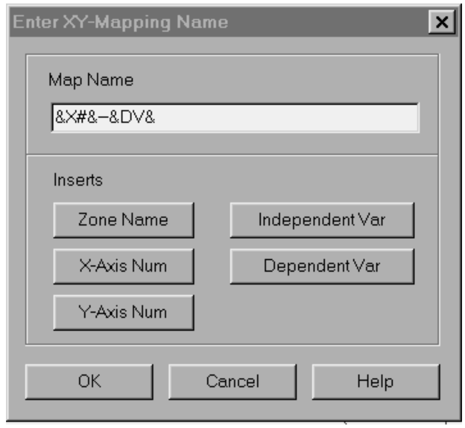



**Figure 8-4.** Assigning XY-mapping names.

To activate or deactivate a mapping or mappings:

1. Select the mapping or mappings you want to activate or deactivate.
2. Click Map Show.
3. Click Activate or Deactivate. (You can also click Map Show then drag to Activate or Deactivate, then release.) You may also choose Show Selected Only, which activates the selected maps and deactivates all other maps.

   Active mappings have the word Yes in the column under Map Show; inactive mappings have the word No.

### 8.3.3. Assigning X- and Y-Variables to XY-Mappings

The choice of X- and Y-variables is the heart of the XY-mapping, so once defined, you are unlikely to change them. However, if you are editing a copy of a mapping to create a new mapping, you probably will change the X- and Y-variables. You can do this at any time, for any mapping, using the X-Axis Variable and Y-Axis Variable buttons on the Define XY-Mappings dialog.

To assign a variable to the X-axis (Y-axis) for a mapping or mappings:

1. Select the mapping or mappings for which you want to change the X-axis or Y-axis variable.
2. Click X-Axis Variable (Y-Axis Variable). The Select Variable dialog appears, containing a drop-down menu of all the current data set's variables.
3. Choose the desired variable from the drop-down menu.

### 8.3.4. Assigning Zones to XY-Mappings

Each XY-mapping is restricted to one zone. If your data set has multiple zones, you can specify which zone a particular mapping is restricted to.

To assign a zone to a mapping or mappings:

1. Select the mapping or mappings which you want to assign to a zone.
2. Click Zone. The Select XY-Mapping Zone dialog appears, containing a drop-down menu of all the current data set's zones.
3. Choose the desired zone.

### 8.3.5. Assigning Axes to XY-Mappings

Tecplot's default active XY-mapping assigns the X-Axis and Y-Axis variables to the X1- and Y1-axes, respectively. You can change these assignments, for both active and inactive mappings, using the Which X-Axis and Which Y-Axis fields on the Define XY-Mappings dialog. Use these fields, respectively, to reassign the X-axis variable and Y-axis variable to different axes.

To change the axis assignments for a mapping or mappings:

1. From the Define XY-Mappings dialog, select the mapping or mappings for which you want to change the X-Axis or Y-Axis assignment.
2. Click Which X-Axis (Y-Axis).
3. Click the desired axis (X1 through X5, Y1 through Y5).

For more information on working with multiple X- and Y-axes, see Section 8.5.3, "Using Multiple X- and Y-Axes."

## 8.4. Altering the Style

The ‘‘style’’ of an XY-plot includes each mapping's color, curve type (line segment, spline, and so forth), line pattern (none, solid, dashed, and so forth), symbol type (none, circle, square, and so forth), symbol size (small, medium, or large), and many other attributes. More broadly, you can think of the style as the total visual look of the plot.

One graph is drawn for each active XY-mapping. For example, if you have two mappings with variables assigned to the Y1-axis, one mapping with a variable assigned to the Y2-axis, and repeat these mappings for each of three zones, Tecplot plots nine (*(2+1)*3*) graphs in a single frame. Each graph may have a different style.

Each graph is composed of one or more map layers. The map layers in XY-plots are Lines, Symbols, Bars (for creating bar charts), and Error Bars. Each graph is given an initial color to distinguish it from others. By default, only the Lines map layer is turned on, so no symbols are plotted. All attributes are assigned on a per-mapping basis; that is, the default colors are assigned to differentiate each mapping. By default, all mappings are assigned square symbols.

Using the four XY Plot Attributes dialogs, (Line Attributes, Symbol Attributes, Bar Chart Attributes, and Error Bar Attributes), you set the style attributes for the lines, bar charts, symbols and error bars of the XY-plots. You can also make many of these changes using the Quick Edit dialog accessible from the sidebar. You can set the style of any mapping independently of all other mappings.

### 8.4.1. Activating and Deactivating Map Layers

Changing map layers is the quickest and most visually striking means of changing your XY-plot style. Switching from a line plot to a symbol plot or a bar chart dramatically alters the style. You can also combine layers to create striking visuals—for example, a curve fit together with a bar chart can clearly outline the trend in the data, as in Figure 8-5.

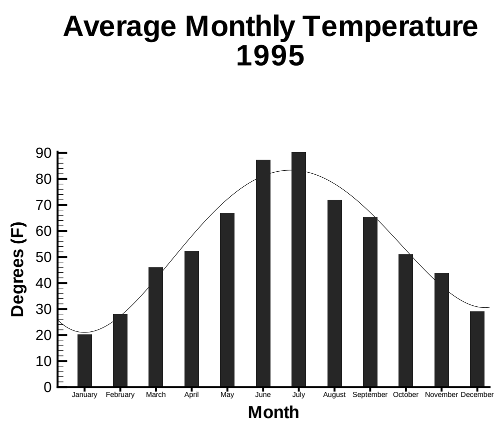



**Figure 8-5.** A bar chart combined with an XY-curve fit.

To activate or deactivate a map layer, click on the map layer's check box on the sidebar. Figure 8-6 shows the map layer area of the sidebar with the Lines and Symbols map layers turned on.

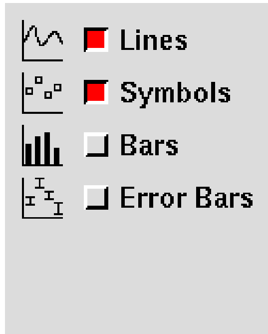



**Figure 8-6.** The map layer area of the sidebar, with Lines and Symbols selected.

## 8.4.2. Altering Line Attributes

Figure 8-7 shows the Line Attributes dialog, with information from the demo file **`slice.plt`**. The first two columns, Map Num and Map Name, list the mapping number and name. The Map Show field shows which mappings are currently active. A mapping must be active for it to be displayed, although an active mapping is not always visible. The remaining columns of the Line Attributes dialog contain specific line attributes, as follows:

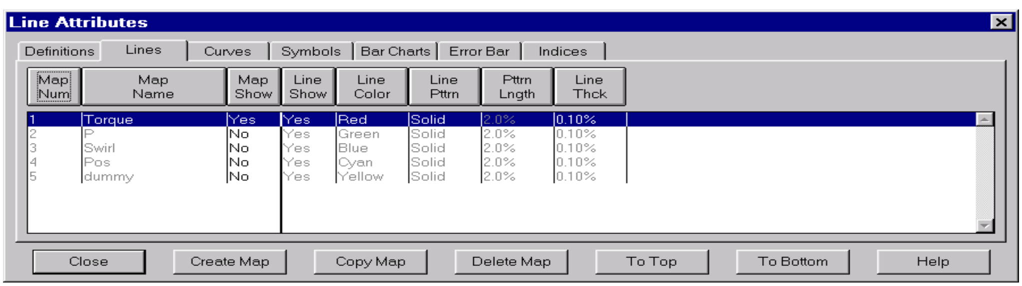



**Figure 8-7.** The Line Attributes dialog.

- **Line Show.**
- **Line Color.**
- **Line Pattern.**

- **Pattern Length.**
- **Line Thickness.**

Figure 8-8 shows the Quick Edit dialog. It allows you to control all of the line attributes. Each of the line attributes is described in detail below.

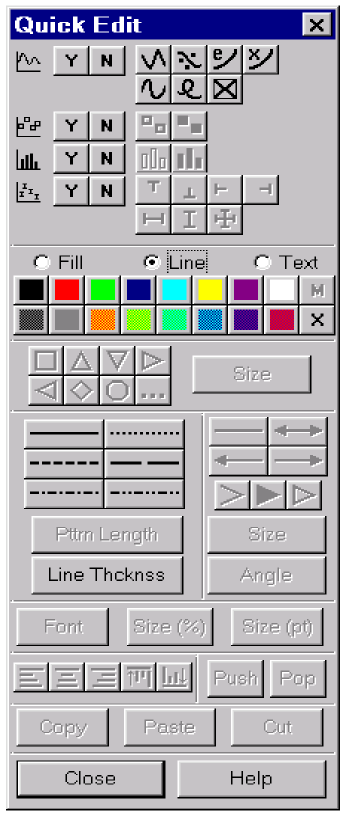



**Figure 8-8.** The Quick Edit dialog for XY-plots.

**8.4.2.1. Choosing Lines to Show.** You can specify whether lines are shown for individual XY-mappings. This option allows you to turn off selected XY-mapping lines, while keeping both the selected XY-mappings and the Lines map layer active. You might want to do this, for example, if you want a line plot of one mapping and a symbol plot of another. In this case, you would set Line Show to “No” for the symbol plot, and Symb Show to “No” for the line plot. See Section 8.4.3.1, “Showing Symbols,” for details on using Symb Show.

To turn the line plot on or off for a mapping or mappings:

1. From the Line Attributes dialog, select the mapping or mappings for which you want to show or hide line plots.
2. Click Line Show.
3. Click Yes to show the line for the selected mappings, No to turn off the line.

or

1. In the workspace, click on the plot line that you want to show or hide.
2. In the Quick Edit dialog, next to the Lines map layer button, click Y to show the line for the selected mappings, N to turn off the line.

**8.4.2.2. Choosing a Line Color.** Set line color for XY-plots using the Line Color drop-down in the Line Attributes dialog, or using the Quick Edit dialog. There is, however, a difference between the two methods. The Quick Edit dialog changes the line color for all selected objects, so that, for example, if there are symbols as well as lines drawn, the symbol color would change with the line color. If you use the Line Color field to change the line color, only the lines change.

To change the line color using the Line Color drop-down in the Line Attributes dialog:

1. From the Line Attributes dialog, select the mapping or mappings for which you want to assign a new color.
2. Click Line Color. A drop-down of Tecplot's basic colors appears.
3. Click the desired color.

To change the line color using the Quick Edit dialog:

1. In the workspace, click on the line whose color you wish to change.
2. On the sidebar, click Quick Edit to call up the Quick Edit dialog, if it is not already on your screen. Figure 8-9 shows the color edit area of the Quick Edit dialog.

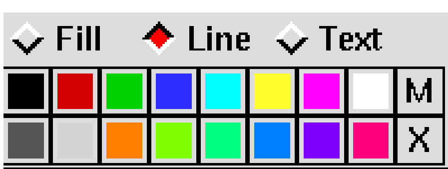



**Figure 8-9.** The color edit area of the Quick Edit dialog.

3. If the Line option button in the color edit area is not already selected, select it.
4. Click on the desired color.

**8.4.2.3. Choosing a Line Pattern.** Set line patterns for XY-line plots using the Line Pttrn drop-down, or the line pattern area of the Quick Edit dialog, shown in Figure 8-10. The choices

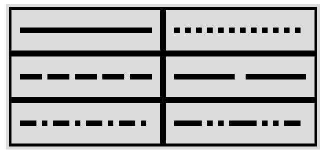


**Figure 8-10.** The line pattern area of the Quick Edit dialog.

are: Solid, Dashed, DashDot, Dotted, LongDash, and DashDotDot. The line pattern setting affects the line or curve drawn for a plot, but not the symbols. Line pattern also has no effect for variables assigned as error bars. Error bars, bar charts, and symbols are always drawn using a solid line pattern.

To change the line pattern for a mapping or mappings from the Line Attributes dialog:

1. From the Line Attributes dialog, select the mapping or mappings whose line pattern you want to change.
2. Click Line Pttrn. A drop-down menu containing the six pattern types appears.
3. Choose the desired pattern.

To change the line pattern using the Quick Edit dialog:

1. In the workspace, click on the line whose line pattern you wish to change.
2. On the sidebar, click Quick Edit to call up the Quick Edit dialog, if it is not already on your screen.
3. In the line pattern area of the dialog (Figure 8-10), click on the desired pattern.

**8.4.2.4. Specifying Pattern Length.** Set the pattern length for patterned lines using either the Pttrn Lngth drop-down menu on the Line Attributes dialog, or the Pttrn Length drop-down menu on the Quick Edit dialog. The pattern length is measured as a percentage of the frame height for one complete cycle of the pattern.

To set the pattern length for a mapping or mappings:

1. Select the mapping or mappings for which you want to change the pattern length. (If you are using the Lines Attribute dialog, select the mappings on that page; if you are using the Quick Edit dialog, select them in the workspace.)
2. Click the appropriate button: Pttrn Lngth on the Line Attributes dialog, or Pttrn Length on the Quick Edit dialog. A drop-down appears containing pre-set choices and an Enter option.
3. Click the desired drop-down option. If you select Enter, an Enter Value dialog appears.
4. (Enter option only) Enter the value for the line pattern length as a percentage of frame height.

**8.4.2.5. Specifying Line Thickness.** Set the thickness of lines using the Line Thck drop-down on the Line Attributes dialog, or the Line Thcknss drop-down menu on the Quick Edit dialog. You can choose from pre-set widths, or enter an arbitrary width as a percentage of the frame height. If you use the Quick Edit dialog, the new line thickness affects all displayed attributes of the selected mappings. For example, it also affects symbol and bar chart line thickness.

To set the line thickness for a mapping or mappings:

1. Select the mapping or mappings for which you want to change the line thickness. (If you are using the Line Attributes dialog, select the mappings on that page; if you are using the Quick Edit dialog, select them in the workspace.)
2. Click the appropriate button: Line Thck on the Line Attributes dialog, or Line Thcknss on the Quick Edit dialog. A drop-down menu appears containing pre-set choices and an Enter option.
3. Click the desired drop-down menu option. If you select Enter, an Enter Value dialog appears.
4. (Enter option only) Enter the value for the line thickness as a percentage of frame height.

### 8.4.3. Altering Symbol Attributes

Figure 8-11 shows the Symbol Attributes dialog. The first two columns list the mapping number and name. The Map Show field shows which mappings are currently active. A mapping must be active for it to be displayed, although an active mapping need not be displayed. The remaining columns of the Symbol Attributes dialog contain specific attributes, as follows:

- **Symb Show.**
- **Symb Shape.**
- **Outline Color.**
- **Fill.**

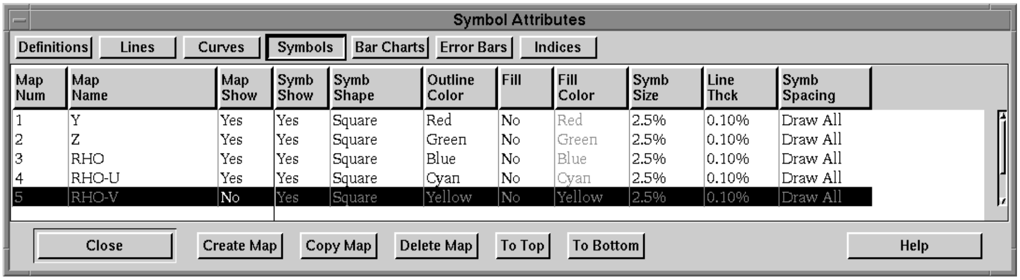



**Figure 8-11.** The Symbol Attributes dialog.

- **Fill Color.**
- **Symb Size.**
- **Line Thck.**
- **Symb Spacing.**

Each of these attributes can also be modified using the Quick Edit dialog, shown in Figure 8-8. Each of these options is described in detail in the sections that follow.

**8.4.3.1. Showing Symbols.** You can specify whether symbols are shown for individual XY-mappings. This option allows you to turn off selected XY-mapping symbols, while keeping both the selected XY-mappings and the Symbols map layer active. You might use this capability, for example, to plot two mappings, one as a line plot, the other as a symbol plot. In this case, you would set Symb Show to “No” for the line plot, and then set the Line Attributes dialog’s Line Show to “No” for the symbol plot. See Section 8.4.2.1, “Choosing Lines to Show,” for procedures for using Line Show.

To turn the symbol plot on or off for a mapping or mappings:

1. From the Symbol Attributes dialog of the Plot Attributes dialogs, select the mapping or mappings for which you want to show or hide symbol plots.
2. Click Symb Show.
3. Click Yes to show the symbols for the selected mappings, No to turn off the symbols.

or

1. In the workspace, click on the symbols of the mapping or mappings for which you want to show or hide symbol plots.
2. On the Quick Edit dialog, in the Symbols layer area, click on Y to show the symbols for the selected mappings, N to turn off the symbols.

**8.4.3.2. Choosing a Symbol Shape.** Use the Symb Shape drop-down on the Symbol Attributes dialog or use the symbol selection area of the Quick Edit dialog, shown in Figure 8-12, to select the symbol type for each mapping.

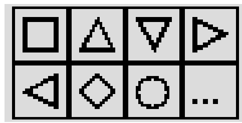


**Figure 8-12.** Symbol selection area of the Quick Edit dialog.

There are seven predefined symbols:

- **Square.**
- **Delta:** An equilateral triangle pointing up.
- **Gradient:** An equilateral triangle pointing down.
- **Right Triangle:** An equilateral triangle pointing to the right.
- **Left Triangle:** An equilateral triangle pointing to the left.
- **Diamond.**
- **Circle.**

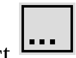
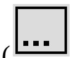
In addition to the predefined symbols, you can choose as a symbol any ASCII character in the following Tecplot fonts: Helvetica-Bold, Math, Greek, User-Defined.

To change the symbol shape using the Symb Shape drop-down in the Symbol Attributes dialog:

1. From the Symbol Attributes dialog, select the mapping or mappings for which you want to assign a new symbol.
2. Click Symb Shape. A drop-down appears containing the seven predefined shapes and an Other option.
3. Click the desired symbol shape, or on Other. If you select Other, the Enter ASCII Character dialog appears.
4. (Other option only) Enter the ASCII character to use as a symbol, and select a font from which to display the symbol.

To change the symbol shape using the Quick Edit dialog:

1. In the workspace, click on the mapping for which you wish to change the symbols.
2. On the sidebar, click Quick Edit to call up the Quick Edit dialog, if it is not already on your screen.

3. In the symbol selection area, click on the desired symbol. If you select , the Enter ASCII Character dialog appears.
4. ( option only). Enter the ASCII character to use as a symbol, and select a font from which to display the symbol.

**8.4.3.3. Choosing a Symbol Outline Color.** Symbols can be filled or unfilled; by default they are unfilled. The symbol's outline color is the color of the unfilled symbol. You can choose an outline color using the Outline Color drop-down menu on the Symbol Attributes dialog or using the color selection area of the Quick Edit dialog. There is a difference between the two methods. The Quick Edit dialog changes the line color for all selected objects. For example, if there are lines as well as symbols drawn, the line color changes as well as the symbol color. If you use the Outline Color field to change the symbol color, only the symbols change.

To change the symbol outline color using the Outline Color drop-down menu in the Symbol Attributes dialog:

1. From the Symbol Attributes dialog, select the mapping or mappings for which you want to assign a new color.
2. Click Outline Color. A drop-down menu of Tecplot's basic colors appears.
3. Click the desired color.

To change the symbol outline color using the Quick Edit dialog:

1. In the workspace, click on the graph for which you wish to change the symbol colors.
2. On the sidebar, click Quick Edit to call up the Quick Edit dialog, if it is not already on your screen. Figure 8-9 shows the color edit area of the Quick Edit dialog.
3. If the Line option in the color edit area is not already selected, select it.
4. Click on the desired color. (Multi-coloring is not an option in XY-plots.)

**8.4.3.4. Choosing Filled or Outline Symbols.** Symbols may be either filled or unfilled. Filled symbols are outlined using the specified outline color, then filled with the specified fill color. You can specify filled symbols either by turning on Fill on the Symbol Attributes dialog, or by specifying a fill color in the color edit area of the Quick Edit dialog. The methods are slightly different, however: the Fill attribute on the Symbol Attributes dialog affects only XY-plotting symbols, while the fill color on Quick Edit affects all selected objects. Thus, you can potentially flood selected geometries as well as plotting symbols when you use the Quick Edit approach.

To choose filled or outline symbols from the Symbol Attributes dialog:

1. Select the mapping or mappings for which you want to choose filled or outline symbols.
2. Click Fill.
3. Click Yes for filled symbols, No for outline symbols.

To choose filled symbols using the Quick Edit dialog, or to turn off fill if you already have filled symbols:

1. In the workspace, click on the graph(s) for which you wish to choose filled symbols.
2. On the sidebar, click Quick Edit to call up the Quick Edit dialog.
3. Click  for filled symbols, or  for hollow symbols.

You can also choose filled symbols by simply selecting a fill color for the selected graphs:

1. Figure 8-9 shows the color edit area of the Quick Edit dialog. If the Fill option button in the color edit area is not already selected, select it.
2. Click the desired color, or click X in the color edit area to turn off fill.

If you have filled symbols, and you select the Line option button in the color edit area, choosing X makes the symbol outline color the same as the symbol fill color.

**8.4.3.5. Choosing a Fill Color.** If you use the Symbol Attributes dialog to specify that a mapping's symbols should be filled, use the Fill Color drop-down menu to select a fill color for filled symbols from among Tecplot's basic colors.

To specify a fill color from the Symbol Attributes dialog:

1. Select the mapping or mappings for which you want to choose a fill color.
2. Click Fill Color. A drop-down menu appears containing Tecplot's basic colors.
3. Click the desired color.

To choose a fill color using the Quick Edit dialog:

1. In the workspace, click on the graph(s) for which you wish to choose filled symbols.
2. On the sidebar, click Quick Edit to call up the Quick Edit dialog.
3. If the Fill option is not already selected, select it. Click on the desired color, or click X in the color edit area to turn off fill.

**8.4.3.6. Choosing a Symbol Size.** Select the symbol size for your XY-plotting symbols using either the Symb Size drop-down menu on the Symbol Attributes dialog or the Size drop-down in the symbol selection area of the Quick Edit dialog.

To specify the symbol size for a mapping or mappings:

1. Select the mapping or mappings for which you want to change the symbol size. (If you are using the Symbol Attributes dialog, select the mappings on that page; if you are using the Quick Edit dialog, select them in the workspace.)
2. Click Symb Size on the Symbol Attributes dialog, or Size on the Quick Edit dialog (be sure you are selecting the Size option in the symbol selection area, immediately to the right of the eight symbol options). A drop-down menu appears containing pre-set choices and an Enter option.
3. Click the desired drop-down menu option. If you select Enter, an Enter Value dialog appears.
4. (Enter option only) Enter the value for the symbol size as a percentage of frame height.

**8.4.3.7. Specify Line Thickness.** To specify the thickness of lines used to draw the plotting symbols, use either the Line Thck drop-down menu on the Symbol Attributes dialog or the Line Thcknss drop-down menu on the Quick Edit dialog. The Line Thck drop-down menu on the Symbol Attributes dialog affects only XY-plotting symbols, while the Line Thcknss drop-down menu on the Quick Edit dialog affects all selected objects.You can choose from pre-set widths, or specify a width as a percentage of the frame height.

To set the line thickness for a mapping or mappings:

1. Select the mapping or mappings for which you want to change the line thickness. If you are using the Symbols Attribute dialog, select the mappings there; if you are using the Quick Edit dialog, select them in the workspace.
2. Click Line Thck on the Symbol Attributes dialog, or Line Thcknss on the Quick Edit dialog. A drop-down menu appears containing pre-set choices and an Enter option.
3. Click the desired drop-down menu option. If you select Enter, an Enter Value dialog appears.
4. (Enter option only) Enter the value for the line thickness as a percentage of frame height.

**8.4.3.8. Specify Symbol Spacing.** To specify the spacing between symbols, use the Symb Spacing drop-down menu on the Symbol Attributes dialog. You can either use one of the drop-down menu’s pre-set values, or enter the spacing as either a percentage of the frame height or by the number of indices to skip. The pre-set values are as follows:

- **Draw All:** All symbols are drawn at every data point.
- **ISkip=2:** Symbols are drawn at every other data point.
- **ISkip=3:** Symbols are drawn at every third data point.
- **ISkip=4:** Symbols are drawn at every fourth data point.
- **Distance=1%:** Symbols are drawn at the first data point and subsequently at data points that are at least one percent of the frame height distant from the previously plotted data point.

- **Distance=2%:** Symbols are drawn at the first data point and subsequently at data points that are at least two percent of the frame height distant from the previously plotted data point.
- **Distance=3%:** Symbols are drawn at the first data point and subsequently at data points that are at least three percent of the frame height distant from the previously plotted data point.

To specify the symbol spacing:

1. From the Symbol Attributes dialog, select the mapping or mappings for which you want to specify the symbol spacing.
2. Click Symb Spacing.
3. Click the desired option. If you select Enter Index or Enter Distance, an Enter Value dialog appears.
4. (Enter Index only) Enter the I-index skip between XY-plot symbols.
5. (Enter Distance only) Enter the distance between XY-plot symbols as a percentage of the frame height.

## 8.5. Controlling the X- and Y-Axes

By default, Tecplot creates XY-plots with one X-axis and one Y-axis. The length of the axes, their ranges, and other axis attributes such as tick marks and labels, are determined for you automatically, but Tecplot allows you to customize these features with almost unlimited flexibility. This section describes how to perform those axis manipulations unique to XY-plots; see Chapter 16, “Controlling Axes,” for a complete description of all axis options.

### 8.5.1. Controlling the Axis Range

Although not unique to XY-plots, controlling the axis range is a very common action in XY-plots. You may want to modify the range, for example, to include additional text or geometries in your axis area, or you may want to have the axes begin and end at round numbers. You control the range of your X- and Y-axes using the Range page of the Edit Axis dialog, shown in Figure 8-13.To call up this page, select Edit from the Axis menu. The default range for both X and Y is the range of the X- and Y-variables. You can alter the range of any active axis. For more details on modifying the axis range, see Section 16.3, “Modifying the Axis Range.”

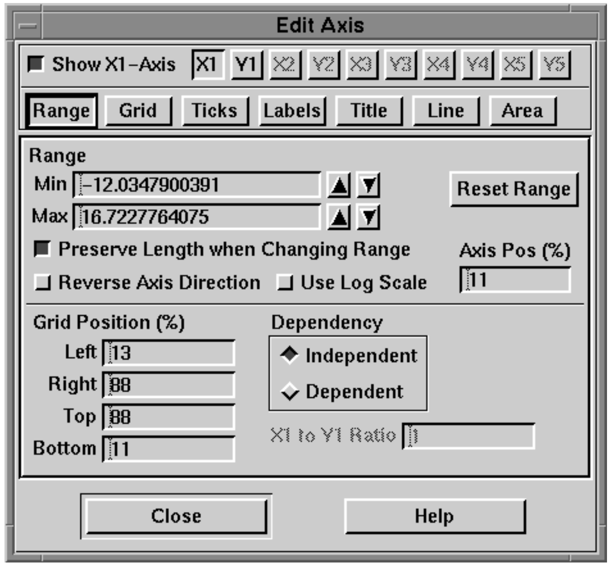



**Figure 8-13.** The Range page of the Edit Axis dialog.

## 8.5.2. Log Axes

You can create a log scale on any or all of the X- and Y-axes. The log axes are available only for XY-plots. When Auto Spacing is selected, large numbers are displayed in scientific notation (that is, $\mathtt{3.48x10^5}$). It is strongly recommended that you use Auto Spacing with log axes.

To specify a log axis:

1. From the Axis menu, choose Edit. The Edit Axis dialog appears.
2. On the Range page of the Edit Axis dialog, select the check box labeled Use Log Scale for each axis you wish to have a log scale.
3. On either the Ticks or Labels page of the Edit Axis dialog, confirm that the check box labeled Auto Spacing is selected (Auto Spacing is turned on by default).

Figure 8-14 shows an XY-plot using a log scale on the Y-axis.

When a log scale is used, polylines and text may be drawn on the plot, but no other grid mode geometries. If you have drawn grid mode circles, squares, rectangles, or ellipses on a plot before selecting the Use Log Scale check box, those geometries will not be drawn when you redraw the plot. However, they remain part of the layout. If you subsequently deselect the Use Log Scale check box, the grid mode circles, squares, and so on reappear.

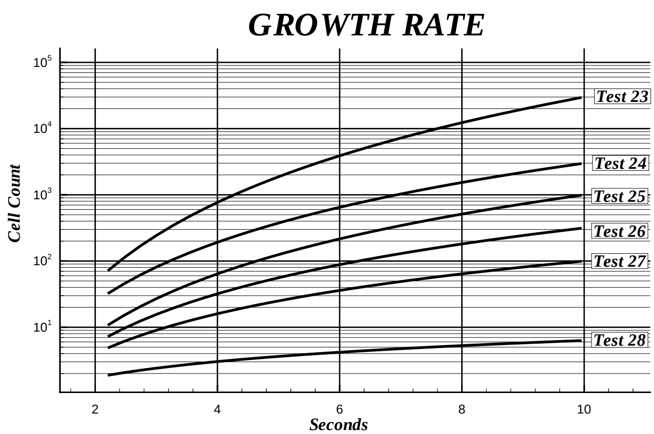



**Figure 8-14.** An XY-plot with a log scale on the Y-axis.

### 8.5.3. Using Multiple X- and Y-Axes

You may want to display XY-mappings which have greatly differing scales on a single plot. For example, you might want to plot a function and its derivative together, or see a plot of both the crime rate and the number of autos per capita versus income level. To create such plots, you assign each mapping to a separate Y-axis; each axis uses the natural scale of the variables assigned to that mapping.

For example, consider the data in the example Tecplot data file **`rain.plt`**; it includes monthly rainfall observations for three U.S. cities, along with two error measurements. The rainfall observations and the error measurements have very different scales. To plot the Seattle rainfall observations and the second error measurement, do the following:

1. Read in the data file **`rain.plt`** from the **`demo/plt`** sub-directory of your Tecplot home directory. Tecplot automatically creates an XY-plot of the Seattle rainfall observations (**`V2`**) versus the Month (**`V1`**).
2. From the XY menu, choose Define XY-Mappings. The Define XY-Mappings dialog appears.
3. Select the mapping named Error 2, then activate it by clicking Map Show and choosing Activate.
4. Click Which Y-Axis, then choose Y2.
5. Redraw to obtain the plot shown in Figure 8-15.

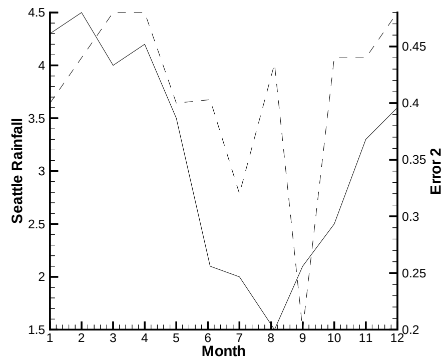



**Figure 8-15.** An XY-plot using two Y-axes.

By default, Tecplot places axis X1 at the bottom of your axis grid area, and subsequent X-axes at the top. Similarly, it places axis Y1 at the left of your axis grid area and subsequent Y-axes at the right. Thus, in Figure 8-15, the Seattle rainfall observations are shown along axis Y1 at the left of the axis grid area, while the error observations are shown along Y2 at the right.

If you have more than two X- or Y-axes, you will typically turn off the display of one or more of the axes to avoid over-plotting the axes.

You do not need to display mappings with different axes at the same time. You may find it convenient to assign different mappings to different axes so that you can set axis ranges, axis positions, or other axis attributes independently for each mapping. Then as you activate and deactivate your mappings, each appears with the settings you have previously set.

Most of the axis settings are axis-specific; you can change these settings for one axis without affecting the corresponding setting on the other axes. The one major exception is the Dependency setting. The Dependency setting affects only X1 and Y1. This, however, may also affect any XY-mappings that use either X1 or Y1 since X1 and Y1 will be more restricted.

## 8.6. Fitting Curves to Data

When you specify a curve type, you control whether Tecplot plots the data and connects the points, or performs a more sophisticated analysis to determine the drawn curve. Curve fitting is an important data analysis tool because it lets you discover hidden trends in seemingly random scatters of data. Curve fits can also help you determine if experimental results match theoretical predictions. Tecplot provides a number of different types of curve fits.

Tecplot offers a variety of curve-fits and spline fits, and they are activated by choosing the appropriate curve type for the XY-mapping. You set the type of curve plotted for an XY-mapping using the Curve Type drop-down on the Curve Attributes dialog, or by using the line type buttons on the Quick Edit dialog. The following settings are available (the names are as shown under the Curve Type drop-down menu; the buttons as shown in the Quick Edit dialog).

### 8.6.1. Curve-Fit Types

Tecplot offers the following types of curve-fits:

- **Line Segments :** Straight line segments that connect adjacent points.
- **Linear Fit:** A straight line is fit to the points using a least-squares algorithm. (Not available on Quick Edit dialog.)
- **Polynomial Fit :** A polynomial curve fit of order *N* (where $1 <= N <= 10$). A polynomial of order *N* is fit to the points using a least-squares algorithm.
- **Exponential Fit :** An exponential curve fit that finds the best curve of the form $Y=e^{b*X+c}$ (equivalent to $Y=a*e^{b*X}$, where a = $e^c$). To use this curve type, Y-values for this variable must be all positive or all negative. If the function dependency is set to *X=f(Y)* then all X-values must be all positive or all negative.
- **Power Fit :** A power curve fit that finds the best curve of the form $Y=e^{b * \ln X + c}$ (equivalent to $Y=a*X^b$, where a = $e^c$). To use this curve type, Y-values for this variable must be all positive or all negative, and the X-values must be all positive. If the function dependency is set to *X=f(Y),* X-values must be all positive or all negative, and the Y-values must all be positive.
- **Spline :** A smooth curve that goes through every point. This curve type assumes that the points represent Y or X values that are the functional values of an independent X- or Y-variable. That is, a spline is drawn through the data points after sorting in increasing values along the independent axis. The spline is a single-valued function of the variable assigned to the independent axis. The spline may be clamped or free. With a clamped spline, you supply the slopes of the curve *(dy/dx)* at each end point; with a non-clamped (natural or free) spline, the slopes are determined for you.

- **ParaSpline :** Same as above, except the assumption is that both X and Y are functions of some other independent variable *s*, where *s* is the accumulated scaled distance between points (that is, *(x,y)=F(s)*). No sorting of the points is performed; the order of the data points from the data file is used. This spline may result in a multi-valued function (of either X and/or Y).
- **Extended :** Select from a list of installed curve-fit add-ons which are extensions to Tecplot. These curve-fits may be provided by Amtec, a third party, or written by users. The functionality of each extended curve-fit is defined by its creator. If you wish to write an extended curve-fit add-on, see the *Add-On Developer's Kit User's Manual* for more information.

Figure 8-16 shows examples of each of Tecplot's curve-fit types.

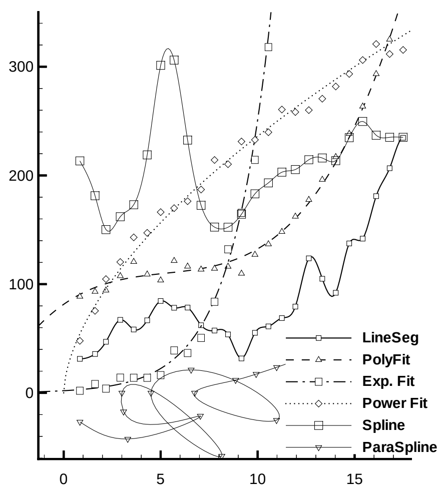



**Figure 8-16.** Tecplot's curve-fit types.

Use the Curve-Fit Attributes dialog, shown in Figure 8-17, to control attributes specific to the curve-fit types. As with the other Plot Attributes dialogs, this page has the mappings listed by number and name, and has the Map Show field for activating and deactivating mappings.

The Curve-Fit Attributes dialog also contains fields for controlling the following attributes:

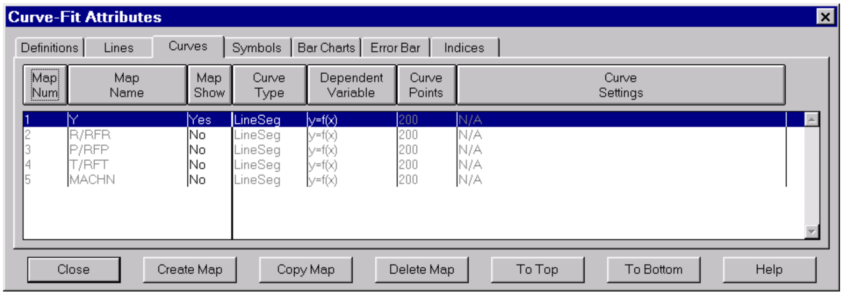



**Figure 8-17.** The Curve-Fit Attributes dialog.

- **Number of curve points:** Users control the maximum number of points displayed for drawing curve fits and splines with the Curve Points field on the Curve-Fit Attributes dialog. Raising the number of points increases not only the accuracy of curve but also the plotting time and the size of print files.
- **Dependent variable:** The choices are *y=f(x)* or *x=f(y)*.
- **Curve settings:** The choices available are dependent upon the curve type.

You may define families of lines in IJ- and IJK-ordered zones to have curve types of splines or curve fits. This means that you can plot the I-varying lines, J-varying lines, or K-varying lines of an IJ- or IJK-ordered zone using these curve types. For example, suppose you create a circular zone with a small number (seven for example) of points around the circumference. A 2-D plot or a straight line segment XY-plot of the J-lines of this zone will show a polygon-shaped zone. Using the Paraspline curve type for the J-lines results in splined mesh lines that reveal the circular shape of the zone. The coefficients used to draw curve fits and splines may be output to a file, as can the actual points used to draw curve fits and splines.

## 8.6.2. Fitting a Straight Line to Your Data

Tecplot fits straight lines to data using the standard least-squares algorithm. It calculates the line for which the sum of the squared differences from the data points to the fitted line is a minimum. A straight line fit implies a linear relationship between the dependent and independent variables. As with all of Tecplot's XY-plotting options, you fit straight lines for particular XY-mappings. For each mapping, you can specify the dependent and independent variables, and, if desired, specify a weighting variable to create a weighted least-squares line. To fit a straight line to your data:

1. From the Curve Attributes dialog, select the mapping or mappings for which you want to fit a straight line.

2. Click Curve Type. A drop-down appears containing all of Tecplot's curve fit and line types.
3. Click Linear Fit.

By default, this option gives a straight line with the X-axis variable as the independent variable and the Y-axis variable as the dependent variable, using no weighting. To choose different dependent and independent variables, see Section 8.6.9, "Assigning Dependent and Independent Variables."

By default, this option fits a line with the X-axis variable as the independent variable and the Y-axis variable as the dependent variable, using no weighting. Use the Curve Fit Settings dialog, shown in Figure 8-18, to specify a different settings:

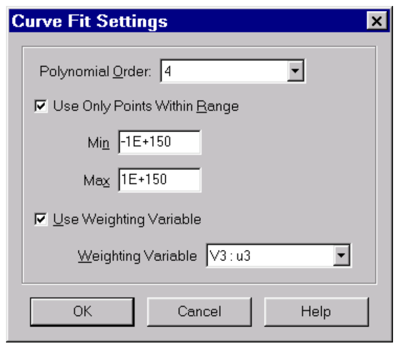



**Figure 8-18.** The Curve Fit Settings dialog.

1. Click Curves to call up the Curve-Fit Attributes dialog.
2. Click Curve Settings. The Curve Fit Settings dialog will appear.
3. To limit the points used in the mapping, select Use Only Points Within Range, and enter minimum and maximum values.
4. To assign a curve-weighting variable, select Use Weighting Variable. Choose your variable from the drop-down menu. For more information on curve weighting, see Section 8.6.10, "Assigning Curve Weighting Variables."

### 8.6.3. Fitting a Polynomial to Your Data

Tecplot fits polynomials to data using the standard least-squares algorithm. That is, it calculates the polynomial of specified order for which the sum of the squared differences from the data points to the fitted polynomial is a minimum. As with all of Tecplot's XY-plotting options,

you fit polynomials for particular XY-mappings. For each mapping, you can specify the dependent and independent variables, and, if desired, specify a weighting variable to create a weighted least-squares line. To fit a polynomial to your data:

1. From the Curve Attributes dialog, select the mapping or mappings for which you want to fit a polynomial.
2. Click Line Type. A drop-down appears containing all of Tecplot's curve fit and line types.
3. Click Polynomial Fit.

You can also create a polynomial by selecting an XY-mapping in the workspace and clicking the  button in the Quick Edit dialog.

By default, this option fits a cubic polynomial with the X-axis variable as the independent variable and the Y-axis variable as the dependent variable, using no weighting. To specify a different settings:

1. Click Curves to call up the Curve-Fit Attributes dialog.
2. Click Curve Settings. The Curve Fit Settings dialog will appear, as shown in Figure 8-18.
3. To change the polynomial order click Poly Order. A drop-down appears containing the integers 1-10. Click the desired polynomial order.
4. To limit the points used in the mapping, select Use Only Points Within Range, and enter minimum and maximum values.
5. To assign a curve-weighting variable, select Use Weighting Variable. Choose your variable from the drop-down menu. For more information on curve weighting, see Section 8.6.10, "Assigning Curve Weighting Variables."

### 8.6.4. Fitting an Exponential Curve to Your Data

If the dependent-variable values are all positive or all negative, and either a preliminary look at the data or prior knowledge gives evidence of an exponential relationship between the variables, you can have Tecplot fit an exponential curve to the data. Tecplot finds the best curve (in the least-squares sense) of the form $Y=e^{b*X+c}$ (equivalent to $Y=a*e^{b*X}$ where $a=e^c$). If $X$ is the dependent variable, the equation is $X=e^{b*Y+c}$. To fit an exponential curve to your data:

1. From the Curve Attributes dialog, select the mapping or mappings for which you want to fit an exponential curve.
2. Click Curve Type. A drop-down appears containing all of Tecplot's curve fit and line types.
3. Click Exponential Fit.

You can also create an exponential fit by selecting an XY-mapping in the Tecplot workspace and clicking  in the Quick Edit dialog.

By default, this option fits an exponential curve with the X-axis variable as the independent variable and the Y-axis variable as the dependent variable, using no weighting. You can change the dependency relationship between the mapping variables using the procedure described in Section 8.6.9, “Assigning Dependent and Independent Variables.”

To change other settings:

1. Click Curve Settings. The Exponential Fit Settings dialog appears, as shown in Figure 8-19.
2. To limit the points used in the mapping, select Use Only Points Within Range, and enter minimum and maximum values.
3. To assign a curve-weighting variable, select Use Weight Variable. Choose your variable from the drop-down menu. For more information on curve weighting, see Section 8.6.10, “Assigning Curve Weighting Variables.”

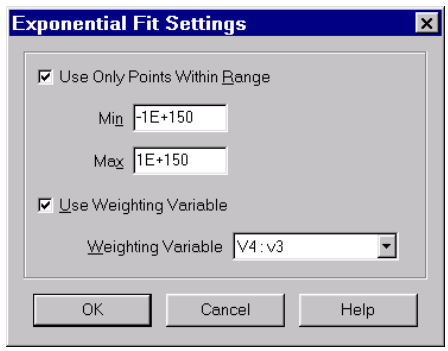



**Figure 8-19.** The Exponential Fit Settings dialog.

## 8.6.5. Fitting a Power Curve to Your Data

If the dependent-variable values are all positive or all negative, and a preliminary look or prior knowledge gives evidence of a power relationship between the variables, you can have Tecplot fit a power curve to the data. Tecplot finds the best curve (in the least-squares sense) of the form $Y=e^{b*lnX+c}$ (equivalent to $Y=a*X^b$ where $a=e^c$). If $X$ is the dependent variable, the equation is $X=e^{b*lnY+c}$. If the independent variable has negative values, they are ignored for purposes of this fit.

To fit a power curve to your data:

1. From the Curve Attributes dialog, select the mapping or mappings for which you want to fit a power curve.
2. Click Curve Type. A drop-down appears containing all of Tecplot’s curve fit and line types.

3. Click Power Fit.

You can also create a power curve fit by selecting an XY-mapping in the workspace and clicking  on the Quick Edit dialog.

By default, this option fits a power curve with the X-axis variable as the independent variable and the Y-axis variable as the dependent variable, using no weighting. You can change the dependency relationship between the mapping variables using the procedure described in Section 8.6.9, “Assigning Dependent and Independent Variables.”

To change other settings:

1. Click Curve Settings. The Power Fit Settings dialog appears, as shown in Figure 8-20.
2. To limit the points used in the mapping, select Use Only Points Within Range, and enter minimum and maximum values.
3. To assign a curve-weighting variable, select Use Weighting Variable. Choose your variable from the drop-down menu. For more information on curve weighting, see Section 8.6.10, “Assigning Curve Weighting Variables.”

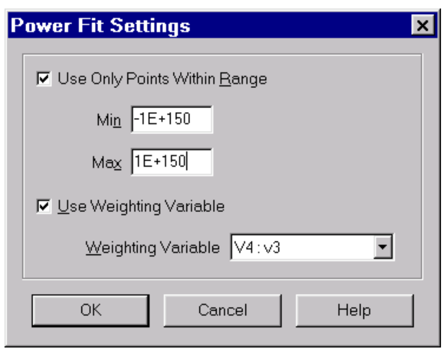



**Figure 8-20.** The Power Fit Settings dialog.

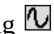
## 8.6.6. Fitting a Spline to Your Data

A spline is a mathematical function defined to link a specified set of points in such a way that the resulting function is continuous and differentiable at each of the specified points. The most common type of spline, the cubic spline, is defined using a set of cubic polynomials, one for each interval between specified points. Splines can be natural or clamped; natural splines are twice-differentiable at the endpoints and the second derivative is zero at those points, while clamped splines need not be twice-differentiable, but have known first-derivatives at the

boundary points. Before plotting the spline, Tecplot sorts the data points in increasing value along the independent axis.

To fit a spline to your data:

1. From the Curve Attributes dialog, select the mapping or mappings for which you want to fit a spline.
2. Click Curve Type. A drop-down appears containing all curve fit and line types.
3. Click Spline.

You can also create a spline fit by selecting an XY-mapping in the workspace and clicking  on the Quick Edit dialog.

By default, this option fits a natural cubic spline with the X-axis variable as the independent variable and the Y-axis variable as the dependent variable, using no weighting.

To specify a clamped spline:

1. Click Curve Settings. The Spline Settings dialog appears, as shown in Figure 8-21.
2. Select Clamp the Spline.
3. If desired, enter new values for Slope at Start and Slope at End.

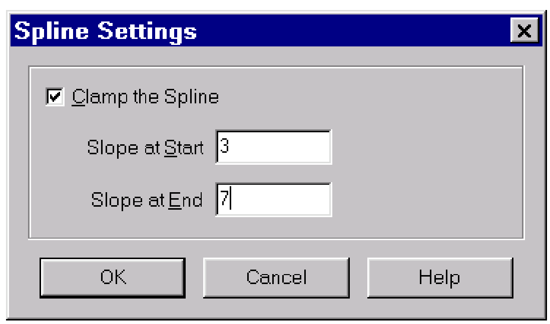



**Figure 8-21.** The Spline Settings dialog.

### 8.6.7. Fitting a Parametric Spline to Your Data

Tecplot's standard spline fit assumes that the spline function is a single-valued function of the specified independent variable. Sometimes, however, you have XY-data that curves back upon itself, but still would like to have a spline-like curve fit to it. Parametric splines solve this problem by assuming that both the X- and Y-values are functions of an underlying variable *s*. The

spline function is then a single-valued function of the underlying variable. Unlike regular splines, parametric splines plot points in their unsorted order, exactly as they are listed in the data set.

To fit a parametric spline to your data:

1. From the Curve Attributes dialog, select the mapping or mappings for which you want to fit a parametric spline.
2. Click Curve Type. A drop-down appears containing all of Tecplot’s curve fit and line types.
3. Click ParaSpline.

You can also create a parametric spline by selecting an XY-mapping in the workspace and clicking  on the Quick Edit dialog.

By default, this option fits a natural cubic spline to the (assumed) underlying variable.

To clamp the spline:

1. Click Curve Settings. The Parametric Spline Settings dialog appears, shown in Figure 8-22.
2. Select Clamp the Spline.
3. If desired, enter new values for Slope at Start and Slope at End.

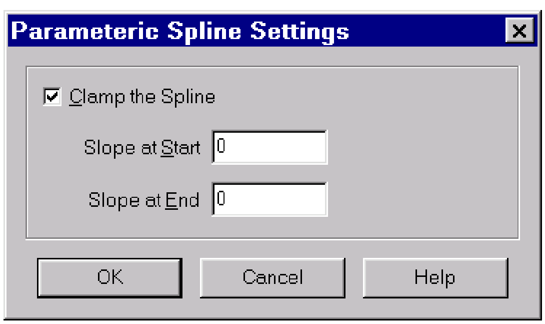



**Figure 8-22.** The Parametric Spline Settings dialog.

### 8.6.8. Fitting an Extended Curve to Your Data

Your XY-data may be plotted using other curve-fits, either supplied with Tecplot, or curve-fits you have added by creating a curve-fit add-on. (For information on creating your own curve-fits, see Chapter 8, “Building Extended Curve-Fit Add-Ons,” in the *Add-on Developer’s Kit User’s Manual*.

To fit an extended curve to your data:

1. Select the mapping or mappings for which you want to use an extended curve-fit from the Curve Attributes dialog.
2. Click Curve Type. A drop-down menu appears containing all of Tecplot's curve-fits.
3. Choose Extended from the drop-down menu. The Choose Extended Curve-Fit dialog will appear with a list of optional curve fits, as shown in Figure 8-23. Select the desired curve-fit.

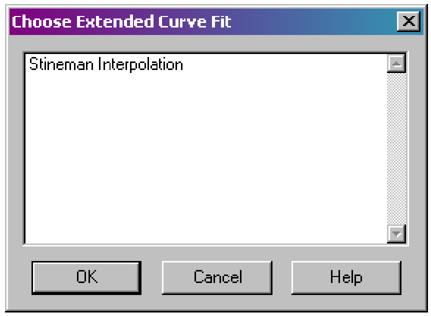



**Figure 8-23.** The Choose Extended Curve-Fit dialog.

You may also invoke an extended curve-fit by selecting an XY-mapping in the workspace, then clicking  on the Quick Edit dialog. This calls up the Choose Extended Curve-Fit dialog, shown in Figure 8-23, from which you may select the desired curve-fit.

Some curve-fits allow optional parameters. These can be accessed by selecting the Curve Settings option on the Curve Attributes dialog. When the curve-fit includes optional settings, a dialog for that curve-fit will appear. For example, the options available for the Stineman Interpolation curve-fit are shown in Figure 8-24.

### 8.6.9. Assigning Dependent and Independent Variables

You specify the dependence relationship between your X-axis and Y-axis variables using the Dependent Variable drop-down on the Curve-Fit Attributes dialog. The default setting is $y=f(x)$; use $x=f(y)$ to use the X-axis variable as the dependent variable and the Y-axis variable as the independent variable. The Dependent Variable has no effect on parametric spline fits.

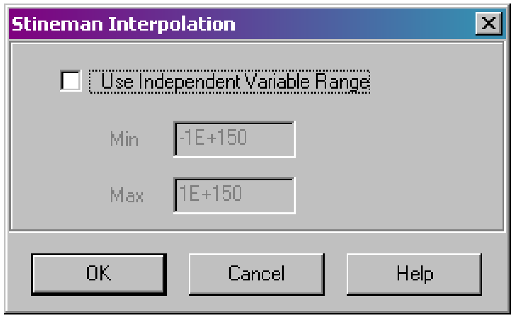



**Figure 8-24.** Stineman Interpolation curve-fit options.

To assign dependent and independent variables:

1. From the Curve-Fit Attributes dialog, select the mapping or mappings for which you want to specify the dependent and independent variables.
2. Click Dependent Variable.
3. Click *y=f(x)* to make the Y-axis variable dependent (the default); click *x=f(y)* to make the X-axis variable dependent.

If you create a horizontal bar chart, Tecplot automatically sets the dependency to *x=f(y).*

## 8.6.10. Assigning Curve-Weighting Variables

You can create weighted curve fits by assigning one variable to be a weighting factor to use when fitting a curve to an XY-mapping. The variable used as the weighting factor is called the curve-weighting variable. Only one curve-weighting variable can be assigned to a given XY-mapping.

When plotting a curve fit, the values in the curve-weighting variable are used to determine a weight factor for each point of the base variable. Relatively higher numbers in the curve-weighting variable mean more significance for a given point. By default, every point is given equal weighting. If the curve-weighting variable is zero at a data point, then that data point has no effect upon the resulting curve. If the curve-weighting variable is two at a data point (and one at all the other points), that point will have twice the effect upon the resulting curve. If the curve-weighting variable is much larger at one point than the others, the weighting of that point will pull the resulting curve close to that point. The weighting coefficients must be integers in the range of 0 to 9,999. Tecplot truncates weighting coefficients defined as floating-point numbers (that is, a weighting coefficient of “**`1.99`**” is truncated to “**`1.0`**”).

For example, consider again the distance-temperature data. There is a small cluster of points centered about Distance=0.1 and Temperature=550. If we add the following weighting variable to the original data file **simpxy.dat**, we can omit this cluster from our analysis:

```
1 1 0 0 0 0 0 0 1 1 1 1 1 1 1 1 1 1 1 1
```

The data file **simpxy2.dat** contains this added variable as variable 3, Weight1.

To assign the curve-weighting variables, see the instructions for Straight Line, Polynomial, Exponential Curve, and Power Curve.

Figure 8-25 shows the weighted linear fit with the cluster of points omitted. For comparison, the original data points and the un-weighted least-squares fit are also plotted. Creating this plot involves the following steps:

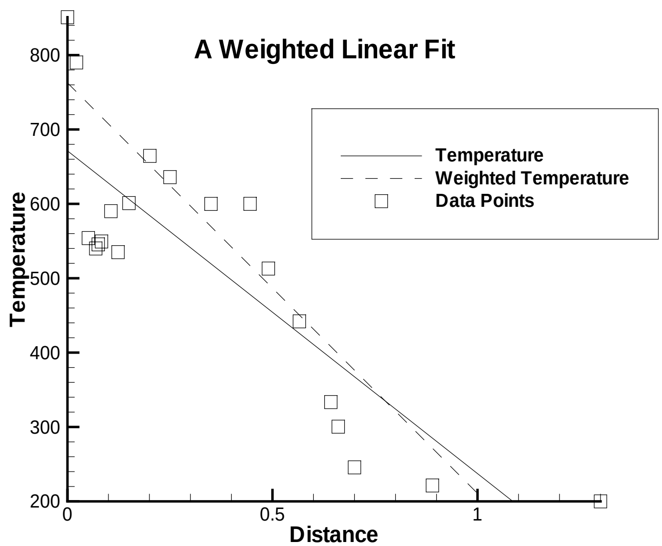



**Figure 8-25.** A weighted linear fit.

1. Read in the data set **simpxy2.dat**, containing the weighting variable **Weight1**.
2. Create two copies of the distance-temperature XY-mapping, and activate the new XY-mappings.
3. Name the first mapping “Temperature,” the second mapping “Weighted Temperature” and the third mapping “Data Points.”

4. Turn on both the Lines and Symbols map layers.
5. Turn off Line Show for the "Data Points" mapping.
6. Set the line pattern for the Weighted Temperature to be dashed, and the curve points for that mapping to be 20.
7. Set the Line Type for Temperature and Weighted Temperature to Linear Fit.
8. Turn off Symb Show for the same two mappings.
9. Assign the Weighting Variable to the Weighted Temperature mapping.
10. Create an XY-plot legend.

### 8.6.11. Extracting Curve Details and Data Points

You can view information about XY-plot curve fits or spline fits in the XY-Plot Curve Information dialog, and write that information to a file for future reference. You can also save the calculated data points along the curve for further analysis in later sessions.

To view the curve details:

1. From the Data menu, select XY-Plot Curve Info. The XY-Plot Curve Information dialog appears, as shown in Figure 8-26.

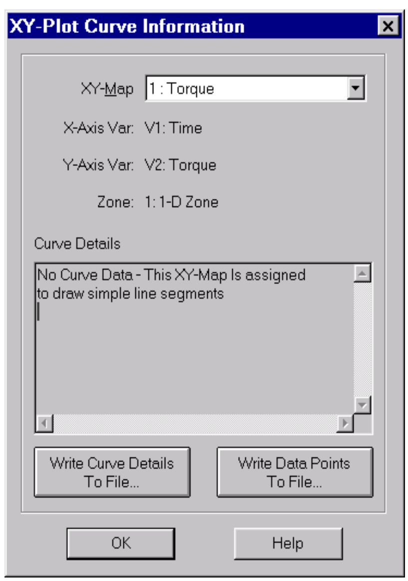



**Figure 8-26.** The XY-Plot Curve Information dialog.

To create an ASCII data file of the points of the curve fits:

1. From the Data menu, choose XY-Plot Curve Info.
2. Select an XY-mapping from the XY-Map drop-down.
3. Click Write Data Points to File. The Write Data Points to File dialog appears.
4. On Motif systems, specify a file name in the text field labeled Selection. On Windows systems, specify a file name in the text field labeled File Name.
5. Click OK.

The data file contains one zone. The zone is I-ordered with the number of points equal to the active curve points setting, set in the Curve-Fit Attributes dialog. The data file has two variables. This is a valid Tecplot ASCII data file that can be read into another frame.

To create an ASCII file with the coefficients for each curve fit or spline:

1. From the Data menu, choose XY-Plot Curve Info. The XY-Plot Curve Information dialog appears.
2. Select an XY-mapping from the XY-Map drop-down.
3. Click Write Curve Details to File. The Write Curve Details to File dialog appears.
4. On Motif systems, specify a file name in the text field labeled Selection.

   On Windows systems, specify a file name in the text field labeled File Name.
5. Click OK.

Depending on the curve type, the coefficient file has one of the forms described in the following sections.

**8.6.11.1. Polynomial Curve Fits.** The coefficient output file for a polynomial least-squares curve fit is of the form:

```
POLYNOMIAL FIT DATA:
 ZONE = Zone-name  (Zone number)
 Y-Axis Variable = Variable-name  (Variable number)
 X-Axis Variable = Variable-name  (Variable number)
 Pwr Coef
 0 C0
 1 C1
 2 C2
 .
 .
 .
 m  Cm
```

The order of the curve fit is *m*, and if the function dependency is *y=f(x)* the curve fit equation is:

$$Y = C_0 + C_1*X + C_2*X^2 + ... + C_m*X^m$$

If the function dependency is *x=f(y)*, the output is as follows:

$$X = C_0 + C_1*Y + C_2*Y^2 + ... + C_m*Y^m$$

**8.6.11.2. Other Curve Fits.** For exponential and power fits, if the function dependency is *y=f(x)*, the coefficient output is as follows:

```
EXPONENTIAL FIT:
ZONE = Zone-name  (Zone number)
Y-Axis Variable = Variable-name  (Variable number)
X-Axis Variable = Variable-name  (Variable number)
Y = e^(B*X+C)

POWER FIT:
ZONE = Zone-name  (Zone number)
Y-Axis Variable = Variable-name  (Variable number)
X-Axis Variable = Variable-name  (Variable number)
Y = e^(B*log(X) + C)
```

If *x=f(y)*, the output is as follows:

```
EXPONENTIAL FIT:
ZONE = Zone-name  (Zone number)
Y-Axis Variable = Variable-name  (Variable number)
X-Axis Variable = Variable-name  (Variable number)
X = e^(B*Y+C)

POWER FIT:
ZONE = Zone-name  (Zone number)
Y-Axis Variable = Variable-name  (Variable number)
X-Axis Variable = Variable-name  (Variable number)
X = e^(B*log(Y) + C)
```

***A*** and ***B*** are replaced with the appropriate values.

**8.6.11.3. Splines.** The coefficient output for a spline has the following form:

```
SPLINE DATA:
ZONE = Zone-name  (Zone number)
```

```
Y-Axis Variable = Variable-name  (Variable number)
X-Axis Variable = Variable-name  (Variable number)

N=001 DIST= x
Coefficients:
  A=value
  B=value
  C=value
  D=value
  .
  .
  .

N=m  DIST=x
Coefficients:
  A=value
  B=value
  C=value
  D=value
```

where *m+1* is the number of XY-points in the zone, and if *y=f(x)*, the following equation is the equation of the polynomial used between points *i* and *i+1*, for *x* between $x_i$ and $x_{i+1}$:

$$Y = A_i + B_i*X + C_i*X^2 + D_i*X^3$$

If the function dependency is *x=f(y)*, then the following equation is the equation of the polynomial used between points *i* and *i+1*, for *y* between $y_i$ and $y_{i+1}$:

$$X = A_i + B_i*Y + C_i*Y^2 + D_i*Y^3$$

**8.6.11.4. Parametric Splines.** For parametric splines, the output coefficients are for two polynomial equations, one for X and one for Y:

```
PARASPLINE DATA:
ZONE = Zone-name  (Zone number)
Y-Axis Variable = Variable-name  (Variable number)
X-Axis Variable = Variable-name  (Variable number)

N=001  DIST=x
XCoefficients:
  A=value
  B=value
  C=value
  D=value
YCoefficients:
  A=value
```

```
    B=value
    C=value
    D=value
    .
    .
    .
  N=m  DIST=x
  XCoefficients:
    A=value
    B=value
    C=value
    D=value
  YCoefficients:
    A=value
    B=value
    C=value
   D=value
```

where *m+1* is the number of XY-points in the zone, and the following equations are equations of polynomials for X and Y between points *i* and *i+1* for *d* between $d_i$ and $d_{i+1}$:

$$X = AX_i + BX_i*d + CX_i*d^2 + DX_i*d^3$$

$$Y = AY_i + BY_i*d + CY_i*d^2 + DY_i*d^3$$

## 8.7. Assigning Error Bars

You can assign one or more variables to be used to compute error bars for another variable using the Error Bar Attributes dialog, shown in Figure 8-27. To access this page, select Error Bar Attributes from the XY menu or click on the Error Bars button from any Plot Attributes dialog. Each error bar variable is associated with a single XY-mapping, so if you want to assign multiple error bar variables to a mapping, you will need to create one copy of the mapping for each error bar you want to assign.

You can use any variable in your data set as an error bar variable, although for them to be meaningful, they should have the same units as the axis along which they are drawn. Sometimes, error variables will be part of your original data file. At other times, you may create error variables using Tecplot's data manipulation utilities. For example, if you know that the values of some measured variable are accurate only to within ten percent, you may create a new variable to use as the error bar variable by multiplying the measured variable by 0.10.

To add error bars to a plot:

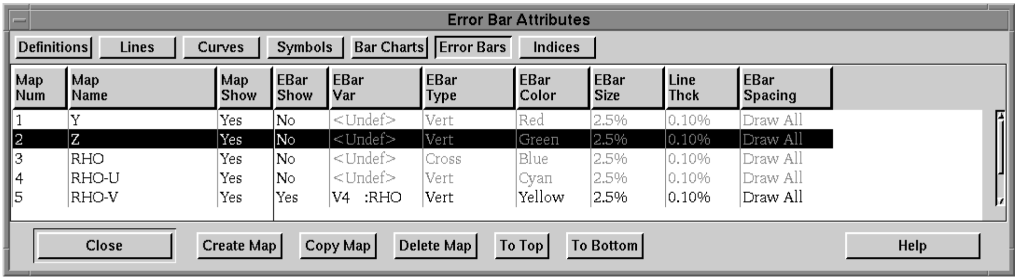



**Figure 8-27.** The Error Bar Attributes dialog.

1. From the Error Bar Attributes dialog, select the mapping or mappings to which you want to assign an error bar variable.
2. Click EBar Var. A Select Variable dialog appears containing a drop-down listing the current data set's variables.
3. Select the desired error bar variable.
4. Click EBar Show, and then on Yes. This enables error bars to be displayed once the Error Bars map layer is activated.
5. On the sidebar, select the Error Bars check box.
6. Click Redraw to regenerate the plot and show the error bars.

### 8.7.1. Selecting an Error Bar Type

Tecplot offers you the choice of seven types of error bars:

- **Top:** Error bar extends upward for positive values (and downward for negative values) of the error-bar variable.
- **Bottom:** Error bar extends downward for positive values (and upward for negative values) of the error-bar variable.
- **Left:** Error bar extends to the left for positive values (and to the right for negative values) of the error-bar variable.
- **Right:** Error bar extends to the right for positive values (and to the left for negative values) of the error-bar variable.
- **Horizontal:** Error bar extends both left and right.
- **Vertical:** Error bar extends both up and down.
- **Cross:** Error bar extends up, down, left, and right.

To select an error-bar type:

1. From the Error Bar Attributes dialog, select the mapping or mappings for which you want to specify the error-bar type.
2. Click EBar Type. A drop-down appears containing the seven error-bar types.
3. Click the desired error-bar type.

When you reverse the direction of an axis using the Reverse Axis Direction option on the Range page of the Edit Axis dialog, the error bars point in the opposite direction. For example, if you reverse the X-axis, Left error bars will point to the right for positive values and to the left for negative values of the error bar variable.

You may assign several variables to be error bars. Each assignment, however, requires a separate XY-mapping. For example, you could assign one variable for the left error bar, copy the XY-mapping, then assign another for the right error bar. You can even assign different variables with the same error bar type. For example, you could assign two variables to be vertical error bars resulting in two vertical error bars at the data point.

An example plot with error bars is shown in Figure 8-28. Variable **`V5`** (called **`ErrP`**) is assigned as a vertical error bar for one mapping. Variable **`V6`** (called **`Err2`**) is assigned as a left error bar for a copy of the same XY-mapping. These vertical and left error bars are plotted on the curve for **`V2`** (called **`RainS`**). The data is in the demo data file **`rain.plt`**.

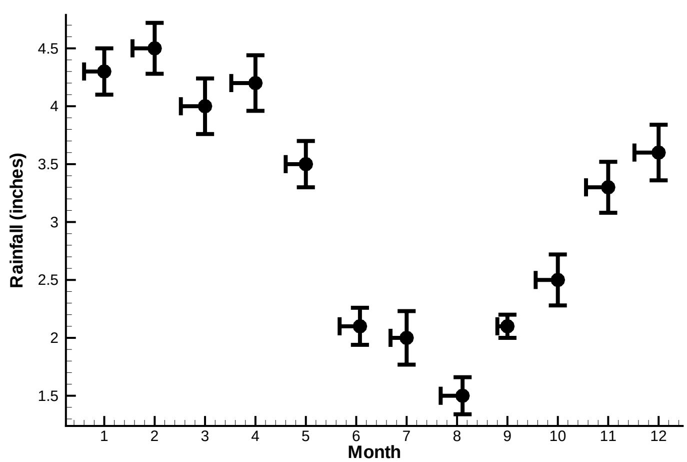



**Figure 8-28.** An XY-symbol plot with error bars.

### 8.7.2. Modifying Other Error Bar Attributes

As with lines and symbols, you can modify most of the attributes with which error bars are drawn—their color, their thickness, their spacing, and the width of the endpoint crossbars. You make all these changes from the Error Bar Attributes dialog.

**8.7.2.1. Choosing an Error Bar Color.** Set error bar line color using the EBar Color drop-down menu on the Error Bar Attributes dialog. To change the line color using the EBar Color drop-down menu:

1. From the Error Bar Attributes dialog, select the mapping or mappings for which you want to assign a new error bar color.
2. Click EBar Color. A drop-down menu of Tecplot's basic colors appears.
3. Click the desired color.

To change the Error Bar color using the Quick Edit dialog:

1. In the workspace, click on the graph for which you wish to change the Error Bar colors.
2. On the sidebar, click Quick Edit to call up the Quick Edit dialog, if it is not already on your screen. Figure 8-9 shows the color edit area of the Quick Edit dialog.
3. If the Line option in the color edit area is not already selected, select it.
4. Click the desired color. (Multi-coloring is not an option in XY-plots.)

**8.7.2.2. Choosing the Crossbar Size.** Select the size of the crossbars on your error bars using the EBar Size drop-down menu on the Error Bar Attributes dialog.

To specify the crossbar size for error bars:

1. Select the mapping or mappings for which you want to change the crossbar size.
2. Click EBar Size. A drop-down menu appears containing pre-set choices and an Enter option.
3. Click the desired drop-down menu option. If you select Enter, an Enter Value dialog appears.
4. (Enter option only) Enter the value for the crossbar size as a percentage of frame height.

**8.7.2.3. Specify Line Thickness.** To specify the thickness of lines used to draw the error bars, use the Line Thck field on the Error Bar Attributes dialog. You can choose from pre-set widths, or enter an arbitrary width as a percentage of the frame height.

To set the line thickness for error bars:

1. Select the mapping or mappings for which you want to change the error bar line thickness.
2. Click Line Thck. A drop-down menu appears containing pre-set choices and an Enter option.

3. Click the desired drop-down menu option. If you select Enter, an Enter Value dialog appears.
4. (Enter option only) Enter the value for the error bar line thickness as a percentage of frame height.

**8.7.2.4. Specify Error Bar Spacing.** To specify the spacing between error bars, use the EBar Spacing drop-down menu on the Error Bar Attributes dialog. You can either use one of the drop-down menu's pre-set values, or enter the spacing as either a percentage of the frame height or by the number of indices to skip. The pre-set values are as follows:

- **Draw All:** All error bars are drawn at every data point.
- **ISkip=2:** Error bars are drawn at every other data point.
- **ISkip=3:** Error bars are drawn at every third data point.
- **ISkip=4:** Error bars are drawn at every fourth data point.
- **Distance=1%:** Error bars are drawn at the first data point and subsequently at data points that are at least one percent of the frame height distant from the previously plotted data point.
- **Distance=2%:** Error bars are drawn at the first data point and subsequently at data points that are at least two percent of the frame height distant from the previously plotted data point.
- **Distance=3%:** Error bars are drawn at the first data point and subsequently at data points that are at least three percent of the frame height distant from the previously plotted data point.

To specify the error bar spacing:

1. From the Error Bar Attributes dialog, select the mapping or mappings for which you want to specify the error bar spacing.
2. Click EBar Spacing.
3. Click the desired option. If you select Enter Index or Enter Distance, an Enter Value dialog appears.
4. (Enter Index only) Enter the I-index skip between error bars.
5. (Enter Distance only) Enter the distance between error bars as a percentage of the frame height.

# 8.8. Creating Bar Charts

A bar chart is an XY-plot that uses vertical or horizontal bars placed along an axis to represent data points. You create bar charts by activating the Bars map layer on the sidebar. You can use the Bars map layer alone to create a pure bar chart, or add symbols, lines, or error bars to create useful visual effects. An example plot with bar charts is shown in Figure 8-5.

To create a pure bar chart:

1. Read in a data file and choose XY frame mode.
2. Select the Bars map layer check box on the sidebar, and deselect the check boxes for all other XY-plot layers.
3. Click Redraw to view the bar chart.

The direction of the bars, vertical or horizontal, is determined by the Depend Variable attribute, which specifies which variable is dependent and which independent (either *y=f(x)* or *x=f(y)*). By default, all mappings are created with Y as the dependent variable, so default mappings will appear as vertical bar charts.

To specify vertical or horizontal bars:

1. From the XY menu, choose Bar Chart Attributes. The Bar Chart Attributes dialog appears, as shown in Figure 8-29.

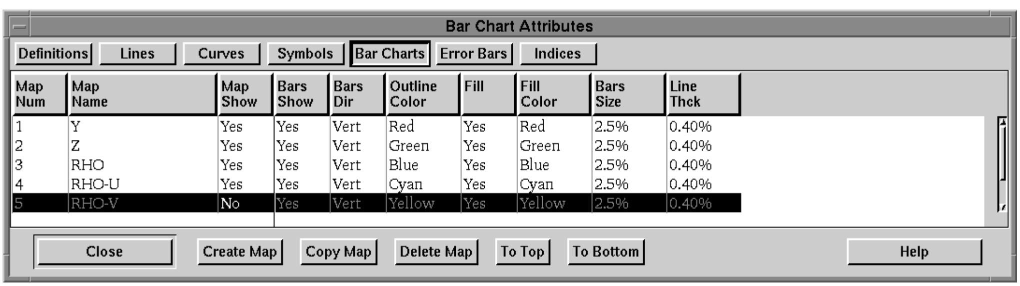



**Figure 8-29.** The Bar Chart Attributes dialog.

2. Click Bars Dir, then choose Horizontal or Vertical. When you change this setting, Tecplot automatically changes the Depend Variable attribute.

To modify other bar chart attributes (Bars Show, Outline Color, Fill, Fill Color, Bar Size, and Line Thickness), use the Bar Chart Attributes dialog, using the same procedures used to set symbol attributes. See Section 8.4.3, “Altering Symbol Attributes,” for details.

## 8.9. Selecting I-, J-, and K-Indices

If your XY-plot data is IJ- or IJK-ordered, an XY-mapping consists of a family of lines; along each line, one index varies while the others are held constant. By default, Tecplot displays the I-varying family of lines. For example, read in the demo data file **`cylinder.plt`** and choose the XY frame mode. You see the family of I-varying lines for Zone 1 of the data, as shown in Figure 8-30.

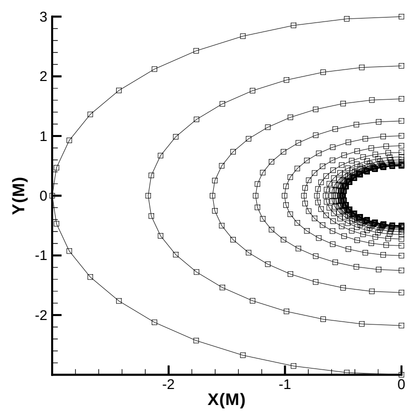



**Figure 8-30.** A family of I-varying lines for the cylinder data.

You can choose whether the I-varying family, the J-varying family, or the K-varying family of lines is drawn using the Index Attributes dialog. You can also choose which members of the family are drawn (and using which data points), by specifying index ranges for each of I, J, and K. The index range for the varying index tells Tecplot which points to include in each line, and the index ranges for the other indices tell Tecplot which lines in the family to include.

To choose the varying index, and thus specify the family of lines to be drawn:

1. From the XY menu, choose Index Attributes. The Index Attributes dialog appears, as shown in Figure 8-31.
2. Select the XY-mapping for which you want to specify the varying index.
3. Click on Varying Index, and choose the desired family (I-varying, J-varying, or K-varying). K-varying is only available if the data is IJK-ordered.

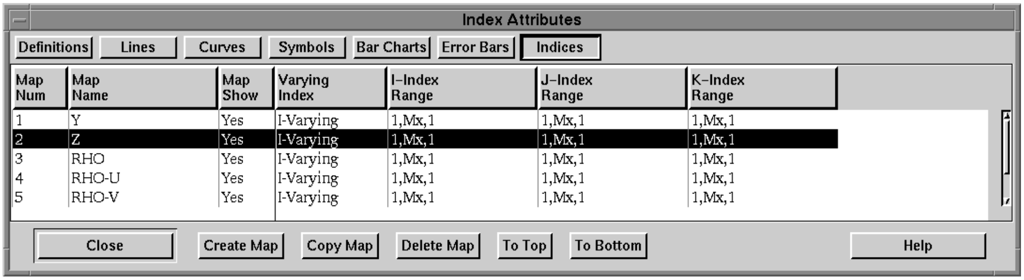



**Figure 8-31.** The Index Attributes dialog.

As a simple example, read in the cylinder data and choose the J-varying index. You obtain the family of J-varying lines shown in Figure 8-32.

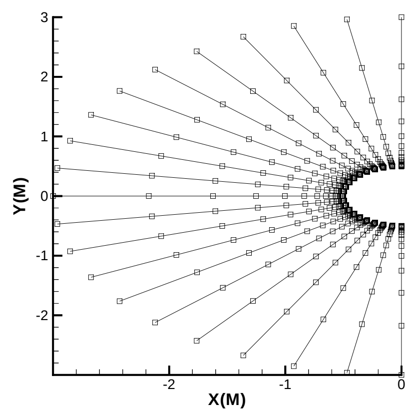



**Figure 8-32.** A family of J-varying lines for the cylinder data.

To specify the index ranges:

1. Select the XY-mapping for which you want to specify an index range.
2. Click on one of I-Index Range, J-Index Range, or K-Index range. The Enter Range dialog appears.

3. Enter a starting index in the Begin field, an ending index in the End field, and a skip factor in the Skip field. A skip of 1 means "use every point in the range," a skip of two means "use every other point," and so on.

For example, for the cylinder data, if you change the I-Index Range to have a skip of three while displaying J-varying lines, you obtain the plot shown in Figure 8-33.

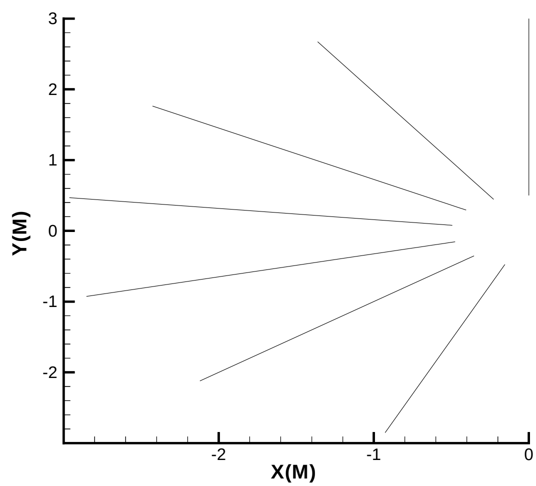



**Figure 8-33.** Family of J-varying lines with I-Index Range having Skip=3.

## 8.10. Adding an XY-Plot Legend

You can generate a legend that shows the line, symbol, and bar chart attributes of all active XY-mappings. This legend can be positioned anywhere on the plot. You can choose to have the mapping names included in the legend.

Tecplot automatically removes redundant entries. That is, if two entries in the legend would look exactly alike, only one is printed.

To create the XY-legend:

1. From the XY menu, choose XY Legend. The XY Legend dialog appears, as shown in Figure 8-32.
2. Select the check box labeled Show XY Legend.
3. If you want the mapping names in the legend, select the check box labeled Show XY-Mapping Names.
4. Format the text for the legend by choosing a color and font, and specifying the text height as a percentage of the frame height. Enter the desired line spacing in the Line Spacing text field.
5. Specify the location of the upper left corner of the legend by entering values in the X (%) and Y (%) text fields. Enter X as a percentage of the frame width and Y as a percentage of the frame height.
6. Select which kind of box you want drawn around the legend (No Box, Filled, or Plain). If you choose Filled or Plain, format the box using the following controls:
   - **Line Thickness:** Specify the line thickness as a percentage of frame height.
   - **Box Color:** Choose a color for the legend box outline.
   - **Fill Color:** (Filled only) Choose a color for the legend box fill.
   - **Margin:** Specify the margin between the legend text and legend box as a percentage of the text height.
7. On the sidebar, click Redraw.

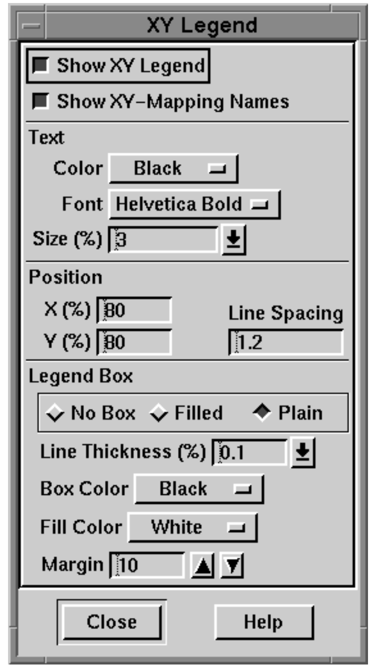



**Figure 8-34.** The XY Legend dialog.

## 8.11. Labeling Data Points

You can label all or some of the data points, or nodes, in your XY-plots with either the index of the data point, the value of the dependent variable at the point, or the pair of values X, Y for the data point.

For example, Figure 8-35 shows an XY-plot with each data point labeled with its XY-value pair.

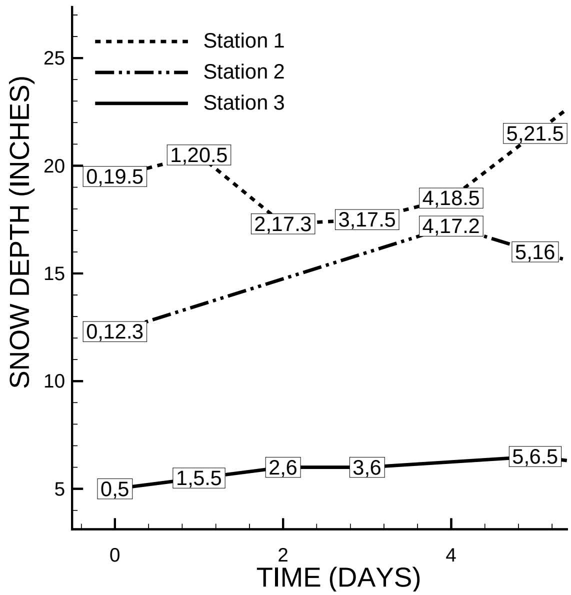



**Figure 8-35.** XY-plot with data labels.

To create data labels:

1. From the Style menu, choose Data Labels. The Data Labels dialog appears, as shown in Figure 8-36.
2. To label the data points, or nodes, select the Show Node Labels check box. If you select this check box, choose one of the three option buttons Show Index Value, Show Dependent Variable Value, or Show X, Y Value Pair.
3. To label all nodes, select the option button labeled By Index in the Skip region. Confirm that the Index Skip text field contains the default value 1. To label only some of the nodes, enter a larger number in the text field. A value of 2 labels every other point, a value of 3 labels every third point, and so on.

   You can also specify label-skipping in terms of distance: simply select the By Distance (%) option button, then enter a value in the adjacent text field. Data points which are closer than the specified distance to the last labeled point are not labeled.
4. Specify the format of the data labels using the following controls:

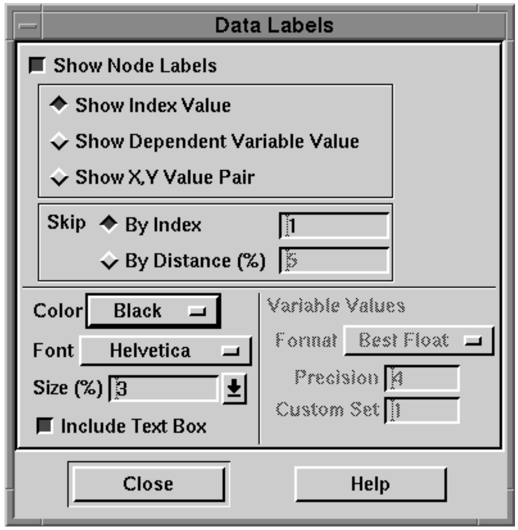



**Figure 8-36.** The Data Labels dialog.

- **Color:** Choose any of Tecplot's basic colors from the color drop-down.
- **Font:** Choose a font from the drop-down.
- **Size(%):** Specify a size for the labels as a percentage of the frame height. Either enter a value or choose a pre-set value.
- **Include Text Box:** Select this check box to include a filled box around each data label.
- **Variable Values:** If you are using a variable value as the data label, you can specify the format of the labels using these controls. The available formats are the same as for tick mark labels; see Section 16.5.3, "Tick Mark Label Formats," for details.

5. Redraw your plot to see the data labels.

# CHAPTER 9 Creating Field Plots

A field plot is any plot created in 2D or 3D frame mode. Such plots combine one or more of the following zone layers:

- **Mesh.**
- **Contour.**
- **Vector.**
- **Scatter.**
- **Shade.**
- **Boundary.**

By default, 2- and 3-D field plots consist of the Mesh and Boundary zone layers. For example, when you read in the data set **`cylinder.plt`** (included with your Tecplot distribution), you automatically see the plot shown in Figure 9-1. The **`cylinder.plt`** data set contains three zones. By default all zones are plotted; Tecplot assigns basic colors cyclically to the individual

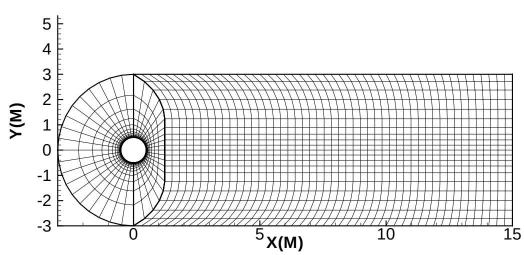



**Figure 9-1.** A 2-D mesh plot.

zones. However, you can assign any of Tecplot's basic colors to each zone. By default, the X- and Y-axes are dependent; if you change the range of one, the other changes to preserve the XY-aspect ratio. You can change virtually all attributes of the plot using the Axis, Field, and Style menus. This chapter discusses the basic plot attributes that are common to all field plots.

## 9.1. Creating 2-D Field Plots

A 2-D field plot typically uses an IJ-ordered or finite-element surface data set. (You can view I-ordered data in a 2-D field plot, but XY-plots are typically more informative. Similarly, you can view IJK-ordered and FE-volume data with 2-D field plots, but 3-D views are usually better.) An IJ-ordered data file has the basic structure shown below:

```
TITLE = "Example: Multi-Zone 2-D Plot"
VARIABLES = "X", "Y", "Press", "Temp", "Vel"
ZONE T="BIG ZONE", I=3, J=3, F=POINT
1.0 2.0 100.0 50.0 1.0
1.0 3.0 95.0 50.0 1.00
1.0 4.0 90.0 50.0 0.90
2.0 2.0 91.0 40.0 0.90
2.0 3.0 85.0 40.0 0.90
2.0 4.0 80.0 40.0 0.80
3.0 2.0 89.0 35.0 0.85
3.0 3.0 83.0 35.0 0.80
3.0 4.0 79.0 35.0 0.80
ZONE T="SMALL ZONE", I=3, J=2, F=POINT
3.0 2.0 89.0 35.0 0.85
3.5 2.0 80.0 35.0 0.85
4.0 2.0 78.0 35.0 0.80
3.0 3.0 83.0 35.0 0.80
3.5 3.0 80.0 35.0 0.85
4.0 3.0 77.0 33.0 0.78
```

This data file has two zones and five variables, and is included with Tecplot as the file **`examples/dat/multzn2d.dat`**. The first zone has nine data points arranged in a three-by-three grid. The second zone has six data points in a three-by-two mesh. Reading this data file yields the mesh plot shown in Figure 9-2.

A 2-D finite-element data file is shown below:

```
TITLE = "Example: 2D Finite-Element Data"
VARIABLES = "X", "Y", "P", "T"
ZONE N=8, E=4, F=FEPOINT, ET=QUADRILATERAL
0.0 1.0 75.0 1.6
1.0 1.0 100.0 1.5
3.0 1.0 300.0 2.0
```

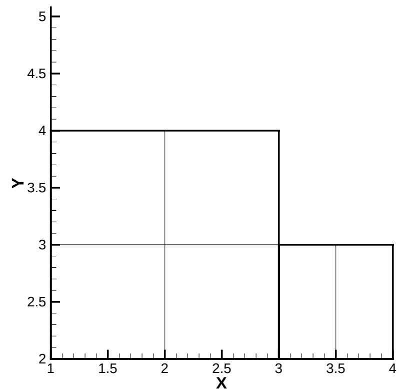



**Figure 9-2.** A 2-D field plot.

```
0.0 0.0 50.0 1.0
1.0 0.0 100.0 1.4
3.0 0.0 200.0 2.2
4.0 0.0 400.0 3.0
2.0 2.0 280.0 1.9
1 2 5 4
2 3 6 5
6 7 3 3
3 2 8 8
```

The above finite-element data file has eight nodes and four elements, with four variables, and is included in your Tecplot distribution as **`examples/dat/2dfed.dat`**. It yields the simple mesh plot shown in Figure 9-3.

## 9.2. Creating a 3-D Field Plot

Creating a 3-D field plot from a 2-D plot is usually as simple as clicking on the button labeled 3D on the sidebar. For example, suppose you read in the file **`spcship.plt`** from the **`demo/plt`** directory, an IJ-ordered data set which by default is displayed in 2-D. Clicking on the 3D button yields the 3-D surface mesh plot shown at the left in Figure 9-4. The spaceship appears on its side by default; you can change this either by rotating the plot around the X-axis or by interchanging the axis assignments of the X- and Y-variables.

To change the axis assignments:

1. From the Axis menu, choose Assign XYZ. The Select Variables dialog appears.
2. For the Y-axis, choose variable Y, and for the Z-axis, choose variable X.

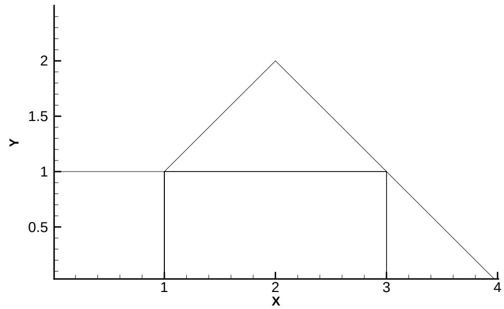



**Figure 9-3.** A 2-D mesh plot of a finite-element data set.

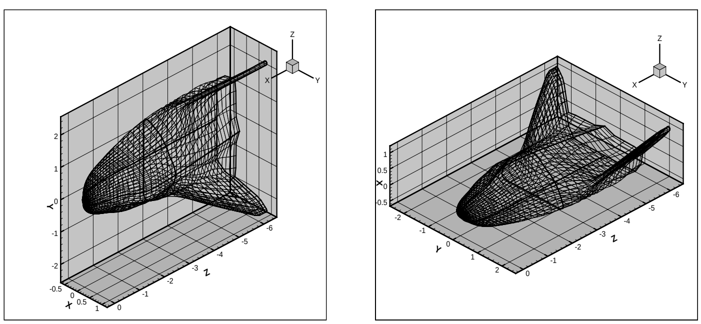


**Figure 9-4.** A 3-D field plot (left). The same plot after interchanging X and Y (right).

**3.** Redraw the plot to obtain the view displayed at the right of Figure 9-4.

For 3-D volume data sets, which include both IJK-ordered and FE-volume data sets, 3D is the default frame mode. That is, when you read in such data sets, Tecplot automatically displays them in 3-D.

IJK-ordered data sets have the general form shown below:

```
TITLE = "Example: Simple 3-D Volume Data"
 VARIABLES = "X", "Y", "Z", "Density"
 ZONE I=3, J=4, K=3, F=POINT
 1.0 2.0 1.1 2.21
 2.0 2.1 1.2 5.05
```

```
 3.0 2.2 1.1 7.16
 1.0 3.0 1.2 3.66
...
```

The complete ASCII data file is included with Tecplot as **`simp3dpt.dat`** (POINT format), and in block format as **`simp3dbk.dat`**. When you read either of these files into Tecplot, you immediately get the plot shown in Figure 9-5.

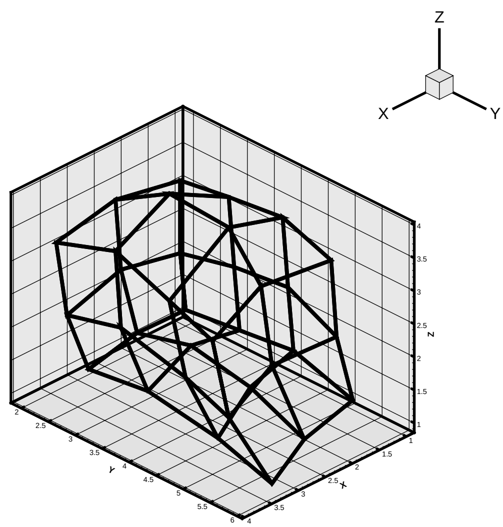



**Figure 9-5.** Plot of a 3-D volume.

Finite-element volume data sets, like FE-surface data sets, consist of two separate lists—the value list and the connectivity list. A portion of a finite-element volume data file is shown below:

```
TITLE = "Example: FE-Volume Brick Data"
 VARIABLES = "X", "Y", "Z", "Temperature"
 ZONE N=14, E=5, F=FEPOINT, ET=BRICK
 0.0 0.0 0.0 9.5
 1.0 1.0 0.0 14.5
 1.0 0.0 0.0 15.0
 1.0 1.0 1.0 16.0
```

```
...
1 1 1 1 2 4 5 3
2 4 5 3 7 10 11 8
4 4 5 5 10 13 14 11
4 4 4 4 9 12 13 10
2 2 4 4 7 6 9 10
```

The full data file, consisting of a single FE-Brick zone, is included with Tecplot as **`febrfep.dat`** (POINT format), and in BLOCK format as **`febrfeb.dat`**. When you read either of these files into Tecplot, you obtain the plot shown in Figure 9-6.

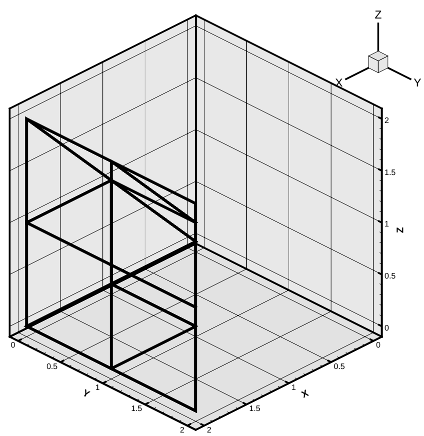



**Figure 9-6.** A 3-D field plot of a finite-element volume data set.

## 9.3. Modifying Your Field Plot

Once you have read in your data, you can modify your field plot attributes using one or more of the Field Plot Attributes dialogs (Mesh Attributes, Contour Attributes, and so forth) or the Quick Edit dialog. No matter which zone layer you are currently modifying, you can control the following attributes from its corresponding Field Plot Attributes dialog:

- Which zones are active.
- Whether the zone layer is visible for each active zone.
- The color. (For the Scatter zone layer, you may specify two colors, an outline color and a fill color.)

A few other attributes are common to some, but not all, zone layers:

- The plot type. (For the Scatter zone layer, this is the symbol shape.)
- The line pattern.
- The pattern length.
- The line thickness.

### 9.3.1. Using Field Plot Attributes Dialogs

The following seven dialogs are listed as Attributes options on the Field menu. We refer to them collectively as the Field Plot Attributes dialogs:

- **Mesh Attributes.**
- **Contour Attributes.**
- **Vector Attributes.**
- **Scatter Attributes.**
- **Shade Attributes.**
- **Boundary Attributes.**
- **Effects Attributes.**
- **Volume Attributes.**

These dialogs allow you to specify plot attributes for each zone layer, zone by zone. On each dialog, the zone information is in the form of a scrolled list. To modify an attribute, the general procedure is the following:

1. Call up the appropriate Attributes dialog via the Field menu or the Details button on the sidebar whenever it is labeled Plot Attributes (the Plot Attributes label appears when no objects are selected and the current mouse mode is either Selector or Adjustor). The Mesh Attributes dialog is shown in Figure 9-7.
2. Select one or more zones, listed in the left hand column of the scrolled list.
3. Click on a column header, which is usually a drop-down showing all the options for that attribute.
4. Choose the desired option to change the attribute for all selected zones.
5. You can quickly move between the different Field Plot Attributes dialogs by clicking the page tabs at the top of the dialog, labeled with an abbreviated form of the various dialog names.

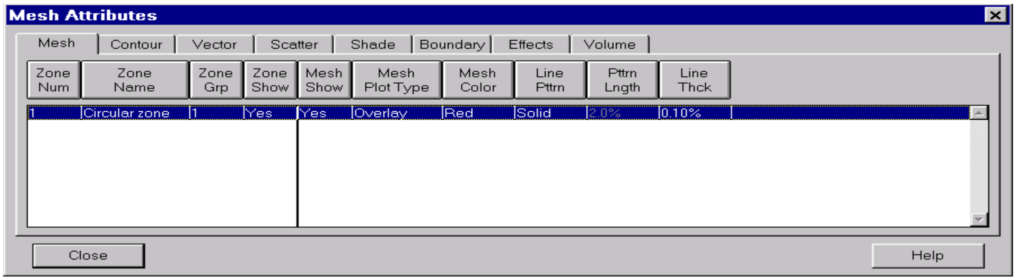



**Figure 9-7.** The Mesh Attributes dialog.

### 9.3.2. Controlling Which Zones are Displayed

By default, all zones are active, meaning capable of being displayed. If a zone is not active, it will not be plotted. At least one zone must be active at all times; if you attempt to deactivate all zones, Tecplot activates the first zone of the group which you tried to deactivate. For example, suppose zones 2 and 3 are active, and you try to turn off both of them. Tecplot automatically turns zone 2 back on. You can activate and deactivate zones from any of the Plot Attributes dialogs.

To activate or deactivate a zone or zones:

1. From any of the Plot Attributes dialogs, select the zone or zones you want to activate or deactivate.
2. Click Zone Show. The choices Activate and Deactivate appear.
3. Click Activate to activate the zones, Deactivate to deactivate the zones, or Show Selected Only to activate the selected zones and deactivate all others.

### 9.3.3. Controlling Zone Layer Display

Whether a given zone layer is displayed for a given zone depends on three things:

- Whether the zone layer is active (controlled via the zone layer buttons on the sidebar).
- Whether the zone is active (controlled in any Attributes dialog).
- Whether the zone is enabled to show the layer (controlled in the relevant zone layer's Attributes dialog).

When you read in a data set, all three conditions are true for all zones and for the Mesh and Boundary zone layers, and the Mesh and Boundary zone layers are shown for all zones. However, you can deactivate certain zones, and for those zones nothing is plotted. You can also

disable individual zone layers for any zone. This is useful if you are creating a complex plot with different plot types for different zones. For example, you might have one zone plotted with contour flooding and another with multi-color mesh lines. In this case, you would selectively turn off the Mesh zone layer in the contour flooded zone and turn off the Contour zone layer in the mesh zone. Both the Mesh and the Contour zone layers would be active globally.

To enable or disable a field layer for a zone or zones:

1. Choose the appropriate Field Plot Attributes dialog.
2. Select the zone or zones for which you want to enable or disable the zone layer.
3. Click on the column header for showing the zone layer (Mesh Show, Contour Show, and so forth). The choices Yes and No appear.
4. Click Yes to enable the zone layer for the selected zones, or No to disable the zone layer for the selected zones.

If you click Yes to enable plotting when the corresponding zone layer is disabled, Tecplot displays a dialog asking if you want to turn on the corresponding zone layer. Click Yes to turn on the zone layer, No to leave it turned off. In either case, the zone has that zone layer enabled, and that zone layer will be displayed at the first Redraw after turning on the corresponding zone layer.

You can also enable or disable each zone layer using the Quick Edit dialog for zones you select interactively in the workspace. Figure 9-8 shows the region of the Quick Edit dialog you use to enable or disable mesh display and also to control the mesh plot type.


**Figure 9-8.** Mesh display and plot type region of the Quick Edit dialog.

To enable or disable a field layer for a zone from the Quick Edit dialog:

1. In the workspace, use the Selector tool to select the zone or zones for which you want to enable or disable a plotting layer.
2. Call up the Quick Edit dialog from the sidebar.
3. Click Y in the appropriate display area to enable the zone layer for the selected zones; click N to disable the zone layer for the selected zones.

### 9.3.4. Choosing Colors

For each zone layer, you can pick a color independently for each zone in the data set. The color chosen for each zone layer is independent of the colors chosen for the other layers. Thus, the mesh color is independent of the colors that you choose for contour lines, vectors, scatter sym-

bols, solid shading, or boundaries. You can choose from any of Tecplot's basic colors, or (for mesh, contour lines, scatter symbols, and vectors) choose the Multi-color option. When you select Multi-color, the zone layer is colored as a function of the contour variable. If no contour variable is currently active, the Contour Variable dialog appears with the default Contour Variable highlighted. You can either select a new contour variable, or click Close to accept the default. A multi-colored plot varies in color like a flooded contour plot—the color of each line segment (between adjacent data points) is determined from the average value of the contour variable at the two data points (together with other options such as the number of contour levels and the current color map).

Specifying a color is essentially the same for all zone layers; to be specific, the procedure for modifying mesh plots is given below. The procedure for changing the color for other zone layers is similar.

To choose a mesh color for a zone or zones:

1. From the Mesh Attributes dialog, select the zone or zones for which you want to specify a color.
2. Click Mesh Color. A drop-down appears containing Tecplot's basic colors and the Multi-color option.
3. Click on the desired color option.

You can also choose the mesh color from the Quick Edit dialog for zones chosen interactively in the workspace. The color region consists of a row of options labeled Fill, Line, and Text, respectively, followed by two rows of color options, one for each of Tecplot's basic colors plus one labeled M for the Multi-color option, and one labeled X, which is not used for mesh plots. When you choose a color from the Quick Edit dialog, the color changes for all visible zone layers for the selected zones.

To choose a mesh color for a zone or zones from the Quick Edit dialog:

1. In the workspace, use the Selector tool to select the zone or zones for which you want to assign a new mesh color.
2. Call up the Quick Edit dialog from the sidebar.
3. Click Line.
4. Click on the desired color option.

### 9.3.5. Choosing a Line Pattern

For mesh plots, contour line plots, and vector plots, you can pick a line pattern independently for each zone in the data set. The line pattern chosen for one layer is independent of the line patterns of the other plot layers. You can choose from any of Tecplot's six line patterns:

- **Solid.**
- **Dashed.**
- **Dash Dot.**
- **Dotted.**
- **Long Dash.**
- **Dash Dot Dot.**

Choosing a line pattern is essentially the same for all zone layers; to be specific, the procedure for modifying mesh plots is given below. The procedure for changing the line pattern for other zone layers is similar.

To choose a mesh line pattern for a zone or zones:

1. From the Mesh Attributes dialog, select the zone or zones for which you want to specify a line pattern.
2. Click Line Pttrn. A drop-down menu appears containing six line pattern types.
3. Click on the desired line pattern option. If you choose any option besides Solid, you can also check the Pattern Length, and modify it as necessary. See Section 8.3.6, “Choosing a Pattern Length.”

You can also choose the line pattern from the Quick Edit dialog for zones chosen interactively in the workspace. Figure 9-9 shows the line pattern region of the Quick Edit dialog, which contains one option for each of Tecplot’s six line patterns. When you choose a line pattern from the Quick Edit dialog, the line pattern changes for all visible layers for the selected zones.

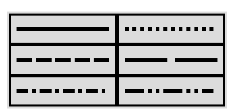
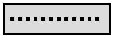
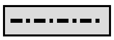


**Figure 9-9.** The line pattern region of the Quick Edit dialog.

To choose a line pattern for a zone or zones from the Quick Edit dialog:

1. In the workspace, use the Selector tool to select the zone or zones for which you want to assign a line pattern.
2. Call up the Quick Edit dialog from the sidebar.
3. Click on the option with the desired line pattern:
   -  Chooses a solid line.

-  Chooses a dotted line.
-  Chooses a dashed line.
-  Chooses a long dashed line.
-  Chooses an alternating dash-and-dot line.
-  Chooses an alternating dash-and-two-dots line.

## 9.3.6. Choosing a Pattern Length

The pattern length determines the cycle length for your line pattern, that is, how long the pattern appears before repeating. In practice, this determines the length of dashed lines, and the spaces between dots and dashes. The pattern length has no effect on solid lines. You specify the pattern length as a percentage of the frame height.

Choosing a pattern length is essentially the same for all zone layers; to be specific, the procedure for modifying mesh plots is given below. The procedure for changing the pattern length for other zone layers is similar.

To choose a mesh pattern length for a zone or zones:

1. From the Mesh Attributes dialog, select the zone or zones for which you want to specify a pattern length.
2. Click Pttrn Lngth. A drop-down menu appears containing five pre-set lengths and an Enter option.
3. Click on the desired option. If you select Enter, an Enter Value dialog appears.
4. (Enter option only) Enter a percentage of the frame height in the Enter Value dialog.

You can also choose the pattern length from the Quick Edit dialog for zones chosen interactively in the workspace. When you choose a pattern length from the Quick Edit dialog, the pattern length changes for all visible zone layers for the selected zones.

To choose a pattern length for a zone or zones from the Quick Edit dialog:

1. In the workspace use the Selector tool to select the zone or zones for which you want to specify the pattern length.
2. Call up the Quick Edit dialog from the sidebar.
3. Click Pttrn Length. A drop-down appears containing five pre-set lengths and an Enter option.
4. Click on the desired option. If you select Enter, an Enter Value dialog appears.
5. (Enter option only) Enter a percentage of the frame height in the Enter Value dialog.

### 9.3.7. Choosing a Line Thickness

For all field layers except Shade zone layer, you can specify a line thickness independently for each zone. The line thickness is independent for each zone layer. The thickness is specified as a percentage of the frame height (see Figure 9-10). Differing line thicknesses can be drawn on the screen, but are not supported on all printers. In particular, HP-GL print files do not support varying line thicknesses. The minimum screen line thickness is one pixel.

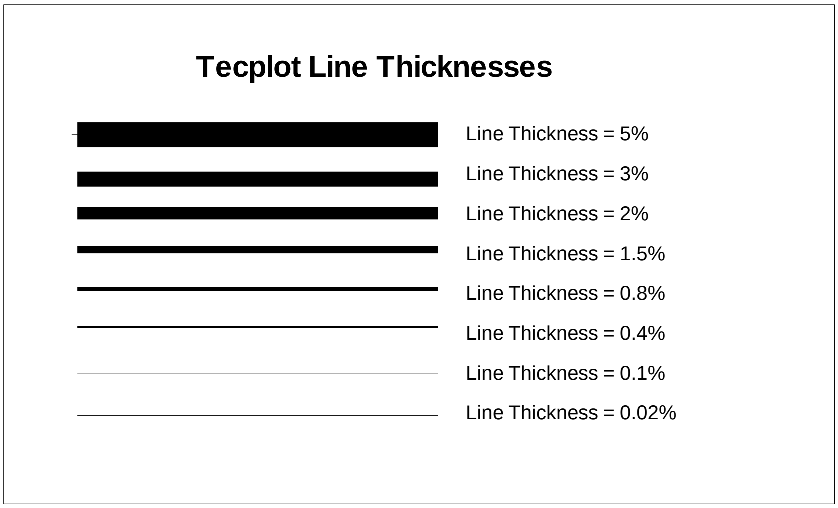



**Figure 9-10.** Varying line thicknesses in Tecplot.

Choosing a line thickness is essentially the same for all zone layers; to be specific, the procedure for modifying mesh plots is given below. The procedure for changing the line thickness for other zone layers is similar.

To choose a mesh line thickness for a zone or zones:

1. From the Mesh Attributes dialog, select the zone or zones for which you want to specify a line thickness.
2. Click Line Thck. A drop-down menu appears containing five pre-set widths and an Enter option.
3. Click on the desired option. If you select Enter, an Enter Value dialog appears.
4. (Enter option only) Enter a percentage of the frame height in the Enter Value dialog.

You can also choose the line thickness from the Quick Edit dialog for zones chosen interactively in the Tecplot workspace. When you choose a line thickness from the Quick Edit dialog, the line thickness changes for all visible zone layers for the selected zones.

To choose a line thickness for a zone or zones from the Quick Edit dialog:

1. In the workspace, use the Selector tool to select the zone or zones for which you want to specify the line thickness.
2. Call up the Quick Edit dialog from the sidebar.
3. Click Line Thcknss. A drop-down menu appears containing five pre-set lengths and an Enter option.
4. Click on the desired option. If you select Enter, an Enter Value dialog appears.
5. (Enter option only) Enter a percentage of the frame height in the Enter Value dialog.

## 9.4. Labeling Data Points and Cells

You can label all or some of the data points, or nodes, in your field plots with either the index value(s) of the data point or the value of some specified variable at each point. You can also label each cell, or element, of the data, with its index (which for finite-element data is its element number).

For example, Figure 9-11 shows a finite-element data set with each node labeled with its node number.

To create data labels:

1. From the Style menu, choose Data Labels. The Data Labels dialog appears, as shown in Figure 9-12.
2. To label the data points, or nodes, select the Show Node Labels check box. If you select this check box, choose one of the two option buttons Show Index Value or Show Variable Value. If you select the Show Variable Value option, choose a variable from the drop-down immediately below the option.
3. To label the cells, or elements, select the Show Cell Labels check box.
4. To label all nodes or cells, confirm that the Index Skip text field contains the default value 1. To label only some of the nodes or cells, enter a larger number in the text field. A value of 2 labels every other point, a value of 3 labels every third point, and so on.
5. Specify the format of the data labels using the following controls:
   - **Color:** Choose any of Tecplot’s basic colors from the color drop-down.
   - **Font:** Choose a font from the drop-down.

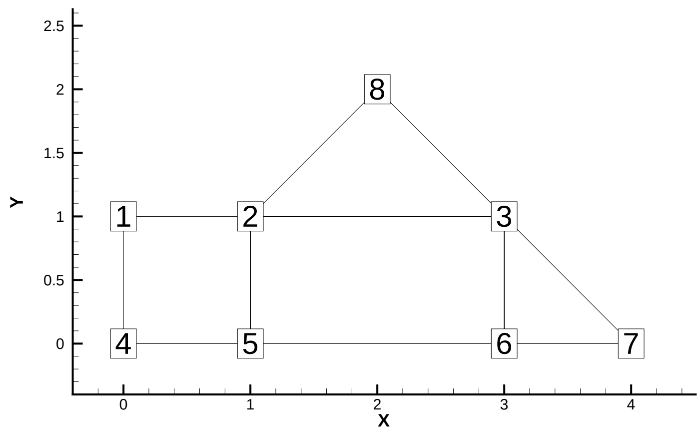



**Figure 9-11.** Finite-element data with data labels.

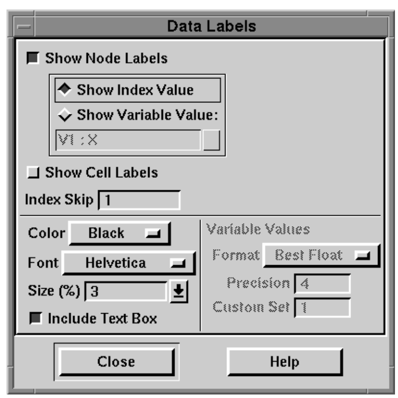



**Figure 9-12.** The Data Labels dialog.

- **Size(%):** Specify a size for the labels as a percentage of the frame height. Either enter a value or choose a pre-set value.
- **Include Text Box:** Select this check box to include a filled box around each data label.
- **Variable Values:** If you are using a variable value as the data label, you can specify the format of the labels using these controls. The available formats are the same as for tick mark labels; see Section 17.5.3, “Tick Mark Label Formats,” for details.

6. Redraw your plot to see the data labels.

## 9.5. 2-D Plotting Order

In 2D frame mode, by default, each zone layer is drawn for all zones before the next layer is drawn. Sometimes, you will want to plot the data zone by zone instead of layer by layer. To do this, choose 2D Draw Order from the Field menu and select the By Zone check box.

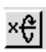
## 9.6. Controlling 3-D Plots

You can view any type of data as a 3-D plot. By default, only IJK-ordered data and finite-element volume data are displayed in 3-D, but you can view other data in 3-D simply by clicking 3D frame mode on the sidebar. Three-dimensional plots can be rotated in space, allowing you to look at your data from any angle. This rotation is probably the most common control you will exercise over your 3-D plots, but Tecplot gives you control over a number of other 3-D plotting attributes that determine precisely how your plot is displayed. This control is necessary because 3-D plots need to provide an illusion of depth in a two-dimensional screen display. The available controls are as follows:

- **3D Rotation:** Control the 3-D orientation of the plot. See Section 9.6.1, “3-D Rotation,” for details.
- **3D View Details:** Set the specifications for a variety of parameters affecting the 3-D display of your plot, including the perspective, field of view, angular orientation of the plot, and view distance. See Section 9.6.2, “3-D View Details,” for details.
- **3D Orientation Axis:** Allows you to control the optional 3-D orientation axis, which shows the current orientation of the three axes. See Section 9.6.6, “3-D Orientation Axis,” for details.
- **3D Reset Axis:** Allows you to reset the 3-D axis sizes and the 3-D origin of rotation. See Section 9.6.7, “3-D Axis Reset,” for details.
- **3D Axis Limits:** Allows you to control the data and axis aspect ratios for 3-D plotting. See Section 9.6.8, “3-D Axis Limits,” for details.
- **3D Light Source:** Control the light source position, as well as the intensity of the light, the background light, and the surface color contrast. See Section 16.1.2, “Lighting,” for more details.
- **Advanced 3D Control:** Specify the default lift fraction for 3-D lines, symbols, and tangent vectors, as well as the 3-D sorting algorithm for the plot. See Section 9.6.4, “3-D Sorting,” for more details.

### 9.6.1. 3-D Rotation

Tecplot allows you to rotate your data in a variety of different ways. Choose one of the six 3-D rotation mouse modes, then drag the pointer in the workspace to rotate your 3-D image. The six rotation mouse modes can be entered by selecting the appropriate sidebar tools, as follows:

- **Spherical** **:** Drag the mouse horizontally to rotate about the Z-axis; drag the mouse vertically to control the tilt of the Z-axis.
- **Rollerball**  **:** Drag the mouse in the direction to move with respect to the current orientation on the screen. In this mode, your mouse acts much like a rollerball.
- **Twist**  **:** Drag the mouse clockwise around the image to rotate the image clockwise. Drag the mouse counterclockwise around the image to rotate the image counterclockwise.
- **X-axis**  **:** Drag the mouse to rotate the image about the X-axis.
- **Y-axis** **:** Drag the mouse to rotate the image about the Y-axis.
- **Z-axis** **:** Drag the mouse to rotate the image about the Z-axis.

Once you have chosen a rotation mouse mode, you can quickly switch to any of the others using the following keyboard shortcuts:

- **s:** Spherical.
- **r:** Rollerball.
- **t:** Twist.
- **x:** X-axis.
- **y:** Y-axis.
- **z:** Z-axis.

### 9.6.2. 3-D View Details

The angular orientation of the plot is defined by three spherical rotation angles:

- $\psi$ **(Psi):** Tilt of eye origin ray away from Z-axis.
- $\theta$ **(Theta):** Rotation of the eye origin ray about the Z-axis.
- $\alpha$ **(Alpha):** Twist about the eye origin ray.

The eye origin ray is a line from the origin of the 3-D object to your eye. The eye origin ray is perpendicular to the plane of the computer screen. These angles define a unique view. These angles are shown in Figure 9-13.

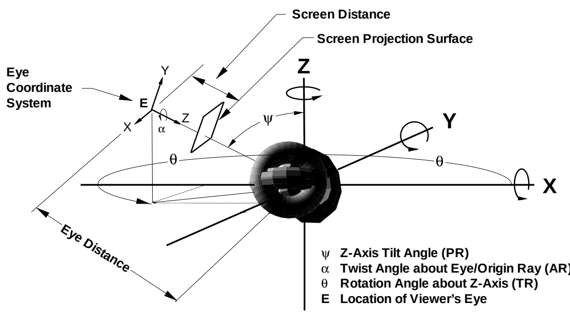



**Figure 9-13.** The 3-D angles and 3-D projection.

When rotating an object, there is a center of rotation about which the rotation takes place. This is called the 3-D origin, and it should not be confused with the actual XYZ-origin of the data. The default 3-D origin is approximately the centroid of all the data in the active zones.

Besides being able to set the 3-D rotate origin via the 3D Rotate dialog, you may also set the origin by positioning the rotation cursor over your data, then pressing the letter O. The rotation origin will then be shifted to the point on the closest surface underneath the cursor.

From the 3D Rotate dialog, you can also choose from four pre-set views or precisely specify the desired orientation of your 3-D plot by entering exact values for the three spherical angles Psi, Theta, and Alpha. You can also define the origin of the 3-D rotation.

**9.6.2.1. Rotate About the Viewer Position.** In addition to the rotation capabilities described above, you may use the Alt key and mouse to rotate about the viewer (instead of rotating the object). Although you may use this feature while in orthographic projection, it is best suited for when perspective projection is being used. The Alt key and your middle mouse button may be used to simulate fly-through type motion. You may move closer to the object using the Alt key and middle mouse button (or Ctrl-Alt-right mouse button), then turn your head using the Alt key and left mouse button.

**9.6.2.2. Rotate Using the 3D Rotate Dialog.** You may also rotate your plots using the 3D Rotate dialog under the View menu, shown in Figure 9-14. At the top of this dialog, there are three options specifying three rotation modes—XYZ-Axis, Spherical, and Rollerball. Depending on the rotation mode chosen, the array of buttons to the right of the options will vary. To rotate the image, click these options. Each click rotates the image by the number of degrees specified in the Rotation Step Size text field.

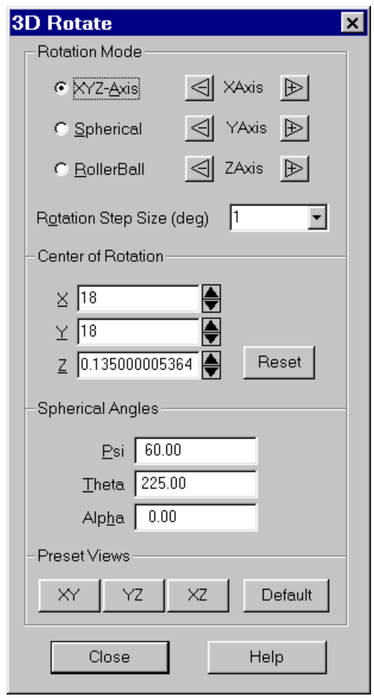



**Figure 9-14.** The 3D Rotate dialog.

## 9.6.3. 3-D Zooming and Translating

Just as in all other plots, you may zoom and translate your plot using the mouse. This may be done using either the Zoom or Translate tools. For most sidebar tools you may also use your middle and right mouse buttons (or Ctrl-right mouse button) to zoom and translate.

When the plot projection is orthographic, zooming with the middle mouse button magnifies the plot. When the plot projection is perspective, zooming with the middle mouse button changes the viewer angle, making the plot appear larger or smaller. If you want to change the viewer's

position by moving closer to or further away from an object hold the Alt key down while using the middle mouse button.

Working with very large data sets may result in slow rotation and translation. If this is the case you may reduce rotations and translations to use a trace of the data instead of the full plot.

To change the rotation and translation drawing behavior:

1. From the sidebar, call up the Display Performance dialog by clicking the Performance button.
2. Click Draw Level for 3D View Changes, then select Trace from the drop-down menu.

If performance remains slow you may also want to turn off the caching of graphics in display lists. This option is also on the Display Performance dialog.

If you are working with multiple frames you may increase performance by choosing to trace all non-current frames. The option is also on the Display Performance dialog.

See Section 31.2, “Customizing Tecplot Interactively,” for further information on customizing the display.

### 9.6.4. 3-D Sorting

For some 3-D plots, Tecplot uses a painter’s algorithm. The data objects are divided into smaller objects. The smallest object is usually a cell, finite-element, vector, or scatter symbol. These objects are sorted based upon the distance from viewer. Tecplot draws the image starting with the objects farthest from the viewer and working forward.

In Tecplot, 3-D sorting occurs whenever you use translucency in a 3-D plot, or whenever you print or print preview a 3-D plot. A quick sorting algorithm is used by default. This does not detect problems such as intersecting objects and is somewhat less accurate. The 3-D sorting for each frame is controlled by the Perform Extra 3D Sorting check box on the Advanced 3D Control dialog, shown in Figure 9-15. If the Perform Extra 3D Sorting check box is selected, a slower, more accurate approach is used to detect problems for you. Call up the Advanced 3D Control dialog by selecting the Field menu’s Advanced 3D Control option.

**Note:** All of the settings in the Advanced 3D Control dialog are specific to the current frame.

There are instances when Tecplot cannot sort correctly. For example, consider elements A, B, and C, where element A overlaps part of element B which overlaps part of element C which overlaps part of element A. Since Tecplot draws only whole elements, one of these elements

will be drawn last and (incorrectly) cover a portion of another element. If this occurs while printing or exporting, choosing an image format will often resolve the problem.

If you specify lift fractions for 3-D lines, tangent vectors, or scatter symbols, plotted objects of the appropriate type are lifted slightly towards you by this fraction so that they lie on top of surface elements. The lift fraction is the fraction of the distance from the 3-D origin of the object to your eye. You may specify lift fractions with the Advanced 3D Controls dialog.

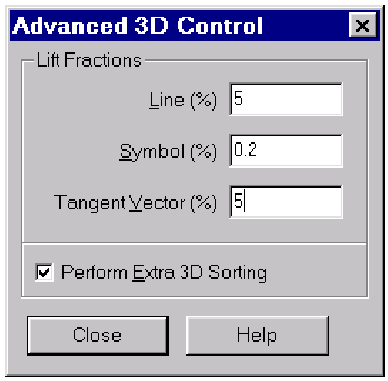



**Figure 9-15.** The Advanced 3D Control dialog.

### 9.6.5. 3-D Projection

The image you see on the screen is a 2-D representation of a 3-D image. Three-dimensional data is projected onto a 2-D plane, your screen. Tecplot offers two projection methods: orthographic and perspective. Orthographic projection is used as the default.

In the 3D View Details dialog, shown in Figure 9-16, you can choose either of the following two types of 3-D projection:

- **Orthographic:** The shape of the objects is independent of distance. This is sometimes an "unrealistic" view, but it is often used for displaying physical objects when preserving the true lengths is important (such as drafting).
- **Perspective:** The shape of the objects is dependent on the field of view angle. The larger the angle the larger the perspective effects. From the 3D View Details dialog you can control the field of view angle and the viewer position. As a convenience you can also change the viewer position by moving closer to or further from the object by changing the view distance.

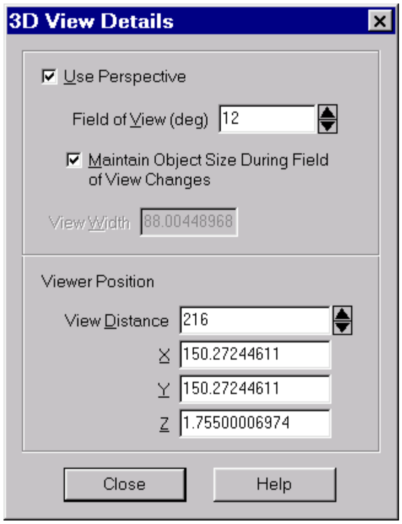



**Figure 9-16.** The 3D View Details dialog.

## 9.6.6. 3-D Orientation Axis

Depending on the view, it may be difficult to determine the current orientation of your 3-D axes. The 3-D orientation axis is a (usually small) representation of your axes that shows you the orientation immediately. By default, all 3-D plots show the 3-D orientation axis in the upper right of the frame. Using the 3D Orientation Axis dialog under the Axis menu, you can control whether the 3-D orientation axis is shown in your plot, and if so, its color, size, line thickness, and the position of the axis origin. You can also position the 3-D orientation axis simply by clicking on it and dragging it to the desired location in the frame.

## 9.6.7. 3-D Axis Reset

By default, the 3-D axes are calculated so that they just surround the data. If you alter your data to expand or contract the overall data size, the axes do not automatically adjust to the new size. For example, if you multiply your X-variable by four, the data will extend four times the length of the X-axis.

You can use the 3D Axis Reset option under the Axis menu to reset the axes so that they once again just surround the data.

The 3D Axis Reset option also resets the 3-D origin, that is, the origin of 3-D rotation. If you have modified your 3-D origin using the 3D Rotate dialog (see Section 9.6.1, “3-D Rotation,” for details), the 3D Axis Reset option will reset it to approximately the centroid of the data.

### 9.6.8. 3-D Axis Limits

In a 3-D plot, whenever you read a data file or manipulate the values of variables assigned to axes or change variables assigned to the axes, Tecplot examines the data and determines how to plot it. The data may require scaling in one or more axis directions, a change of the axis dependency, an adjustment of the space between the data and the axis box, and/or an adjustment of the shape of the axis box.

For example, suppose you read into Tecplot X-Y-Z data that defined a pencil: a long and thin shape. You may want Tecplot to plot this true to scale (that is, a scale factor of one for each axis) in the dependent axis mode, with a long, thin axis box adapted closely to the pencil. Or you may want the data plotted true to scale, but with an axis box nearly cubic in shape. Or perhaps you want the data scaled so the pencil actually appears short and stubby and fills a nearly cube-shaped axis box. Tecplot can plot all of these variations.

Because there are many valid forms in which the data could be plotted, Tecplot requires some user input to determine how to automatically configure the plot the way you want. There are several parameters used by Tecplot to determine when and how the axes are rescaled and resized which are described below. These parameters make up the 3-D axis limit options described below.

You control the allowable shape of your data and axes using the 3D Axis Limits dialog, shown in Figure 9-17. From this dialog accessed from the Axis menu, you can set an aspect ratio and reset limits for both your data and your axes.

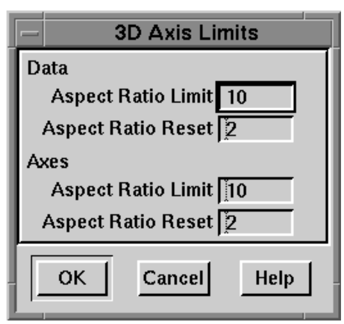



**Figure 9-17.** The 3D Axis Limits dialog.

The data aspect ratio is the ratio of the range of the variable assigned to one axis (multiplied by the axis size factor), divided by the range of the variable assigned to another axis (multiplied by that axis size factor).

For example, if the variable assigned to the X-axis ranged from -2 to 2 (range of four), the X-axis scale factor was one, the variable assigned to the Y-axis ranged from 100 to 500 (range of 400), and the Y-axis scale factor was 0.2, the data aspect ratio would be 20, which is [400*0.2]/[4*1].

The Data Aspect Ratio Limit is the ratio used when Tecplot is automatically resetting data. When the data aspect ratio of any two axes exceeds the Data Aspect Ratio Limit, Tecplot automatically rescales the longer axis (that which has the larger value of range multiplied by scale factor) so that the new data aspect ratio is equal to the Data Aspect Ratio Reset value.

If your plots are usually unscaled, such as plots of real physical objects, you should set the data aspect ratio maximum to a large number like 30. This allows you to plot data that is thirty times longer than it is wide without Tecplot automatically resizing it. If your plots are usually scaled, then you should set the data aspect ratio to a small number like two.

The Data Aspect Ratio Reset value should be equal to or smaller than the data aspect ratio limit. For scaled plots, a reset value of one is reasonable, making the two scale axes equal in length when automatic rescaling takes place.

The Axes Aspect Ratio Limit works like the Data Aspect Ratio Limit, except that it deals with the shape and size of the axes box. For example, if you are viewing a long, slender, physical object, you may want a high data-aspect-ratio limit to keep from rescaling the physical object, but you might also want a low axis-aspect-ratio limit to keep the axes box in a reasonable shape.

The Axes Aspect Ratio Reset value works like Data Aspect Ratio Reset, except that it deals with the shape and size of the axes box. This is the ratio used when Tecplot is automatically resetting the axes box. When the data aspect ratio of any two axes exceeds the Data Aspect Ratio Limit, Tecplot also automatically rescales the longer axis of the axes box (which has the larger value of range multiplied by scale factor) so that the new axes box aspect ratio is equal to the Axes Aspect Ratio Reset value.

Sometimes maintaining the same size factor for each axis is not desirable, especially if the 3-D data do not represent a real physical object. For example, a carpet plot is a 3-D surface where the height (Z) is a single-valued function of width (X) and depth (Y). The data points are usually arranged in an IJ-ordered rectangular array (although this is not required by Tecplot). The variable assigned to the Z-axis could have a range of values that differs greatly from the ranges on the X- and Y-axes. If you plotted this data with equal size factors for each of the three axes, the plot could be long and thin (or very flat). Usually the desired axes for carpet plots are nearly equal in length, but with different size factors. You can have Tecplot automatically rescale the axes for you by changing the Data Aspect Ratio Limit.

# CHAPTER 10 Creating Mesh Plots and Boundary Plots

A mesh plot is a field plot of the lines connecting neighboring data points within a zone. For I-ordered data, the mesh is a single line connecting all of the points in order of increasing I-index. For IJ-ordered data, the mesh consists of two families of lines connecting adjacent data points of increasing I-index and increasing J-index. For IJK-ordered data, the mesh consists of three families of lines, one connecting points of increasing I-index, one connecting points of increasing J-index, and one connecting points of increasing K-index. By changing the zone's Surfaces to Plot on the Volume Attributes dialog, you may limit the plotted mesh to the exterior surface (exposed cell faces) or to selected I-, J-, and K-grid planes. For finite-element zones, the mesh is a plot of all edges of all elements which are defined by the connectivity list for the node points. Mesh lines are straight lines between adjacent points.

A boundary plot is a field plot of the boundaries of ordered-data zones or 2-D finite-element zones. Boundary plots are frequently combined with other plot types.

Because the default field plot type consists of the Mesh and Boundary layers, we have already seen numerous examples of these plots in Chapter 9, "Creating Field Plots." This chapter concentrates on those aspects of mesh and boundary plots unique to those plot types.

## 10.1. Modifying Your Mesh Plot

Once you have read in your data, you can modify your mesh plot attributes using either the Mesh Attributes dialog or the Quick Edit dialog. You can control any of the following attributes from the Mesh Attributes dialog, shown in Figure 10-1:

- Which zones are active. See Section 9.3.2, "Controlling Which Zones are Displayed."
- Whether the mesh is visible for each active zone. See Section 9.3.3, "Controlling Zone Layer Display."

- The mesh plot type. See Section 10.2, “Choosing a Mesh Plot Type,” below.
- The mesh color. See Section 9.3.4, “Choosing Colors.”
- The mesh line pattern. See Section 9.3.5, “Choosing a Line Pattern.”
- The mesh line pattern length. See Section 9.3.6, “Choosing a Pattern Length.”
- The mesh line thickness. See Section 9.3.7, “Choosing a Line Thickness.”

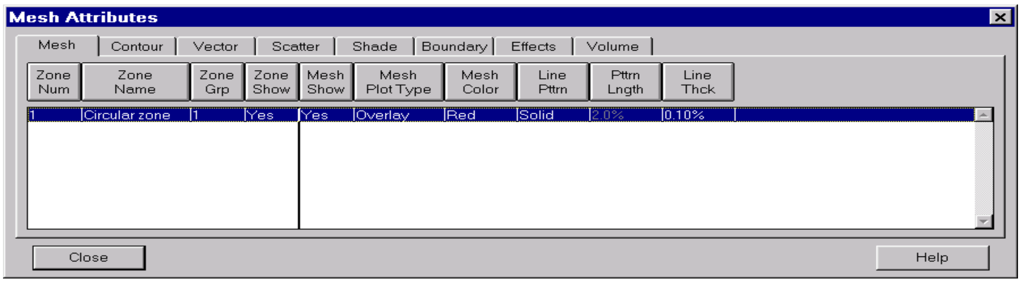



**Figure 10-1.** The Mesh Attributes dialog.

## 10.2. Choosing a Mesh Plot Type

Tecplot has three distinct mesh plot types:

- **Wire Frame:** Mesh lines are drawn underlying all other field plots. Wire frame meshes are drawn below any other field plots (such as contours or vectors) on the same zone. In 3D frame mode, no hidden lines are removed. For 3-D volume zones (finite-element volume or IJK-ordered), the full 3-D mesh consisting of all the connecting lines between data points is not generally drawn because the sheer number of lines would make it confusing. The mesh drawn will depend upon your choice of Surfaces to Plot on the Volume Attributes dialog. See Section 20.1, “Choosing Which Surfaces to Plot,” for further details. By default, only the mesh on exposed cell faces is shown.
- **Overlay:** Like Wire Frame, except that mesh lines are drawn over all other field-plot types except vectors and scatter symbols. For example, if you have flooded contours on a zone, you do not see a wire frame mesh (which would be underneath the contour flooding), but you can see an overlay mesh. In 3D frame mode, the area behind the cells of the plot is still visible (unless some other plot type such as contour flooding prevents this). As with Wire Frame, the mesh drawn is dependent upon your choice of Surfaces to Plot in the Volume Attributes dialog. See Section 20.1, “Choosing Which Surfaces to Plot,” for further details.

- **Hidden Line:** Like Overlay above, except hidden lines are removed from behind the mesh. In effect, the cells (elements) of the mesh are opaque. Surfaces and lines that are hidden behind another surface are removed from the plot. For 3-D volume zones, using this plot type obscures everything inside the zone. If you choose this option for 3-D volume zones using the Volume Attributes dialog's Surfaces to Plot option set to Every Surface, you may want to use a different Surfaces to Plot option (Exposed Cell Faces has the same effect) or use a wire frame mesh instead. If this option is set and the frame mode is not 3D, you get a mesh identical to that for Overlay. The opaque surfaces crated by Hidden Line are not affected by the Lighting zone effect (there is no light source shading).

Figure 10-2 shows the available mesh plot types, along with the effects of choosing Overlay and Wire Frame in combination with contour flooding.

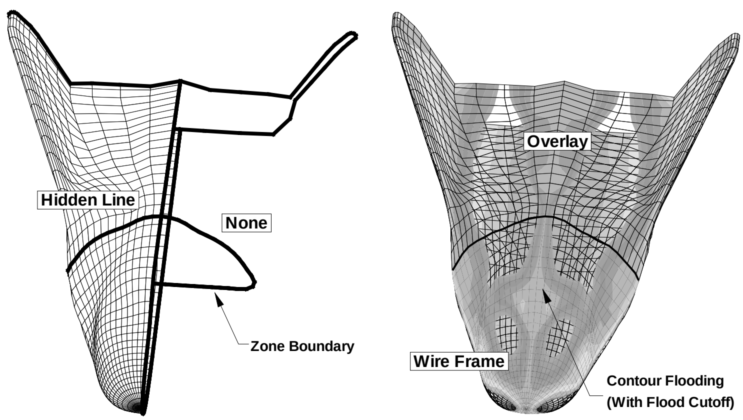



**Figure 10-2.** Mesh plot types.

To choose a mesh plot type for a zone or zones from the Mesh Attributes dialog:

1. From the Mesh Attributes dialog, select the zone or zones for which you want to specify a plot type.
2. Click Mesh Plottype. The options Wire Frame, Overlay, and Hidden Line appear.
3. Click on the desired plot type.

You can also choose the mesh plot type from the Quick Edit dialog for zones chosen interactively in the workspace.

To choose a mesh plot type for a zone or zones from the Quick Edit dialog:

1. In the workspace, use the Selector tool to select the zone or zones for which you want to specify a plot type.
2. Call up the Quick Edit dialog from the sidebar.
3. Click in the Mesh display area for a wire frame plot, click for an overlay plot, or click for a hidden line plot.

## 10.3. Modifying Boundary Plots

The Boundary layer controls the boundary lines for zones. Zone boundaries exist only for ordered zones, or 2-D finite-element zones. They appear as lines around the edges of the zone. Figure 10-3 shows a boundary plot of multiple zones.

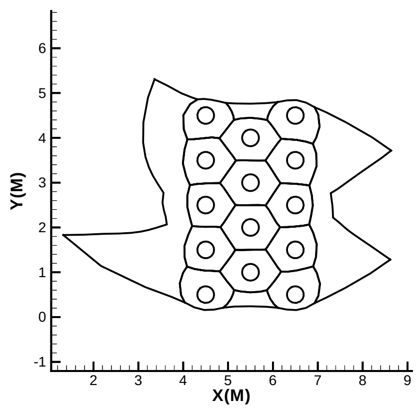



**Figure 10-3.** The boundary of a multiple-zone data set.

Three-dimensional finite-element zones do not have boundaries, although you may use the Extract FE Boundary dialog to create a zone that is the outer boundary or surface of a finite-element zone. See Section 20.4, “Extracting Boundaries of Finite-Element Zones,” for details.

You can control any of the following attributes from the Boundary Attributes dialog, shown in Figure 10-4:

- Which zones are active. See Section 9.3.2, “Controlling Which Zones are Displayed.”
- Whether the boundary is visible for each active zone. See Section 9.3.3, “Controlling Zone Layer Display.”
- Which boundaries are displayed for each zone. See Section 10.4, “Specifying Which Boundaries are Displayed,” below.
- The boundary color for each zone. See Section 9.3.4, “Choosing Colors.”
- The boundary line thickness for each zone. See Section 9.3.7, “Choosing a Line Thickness.”

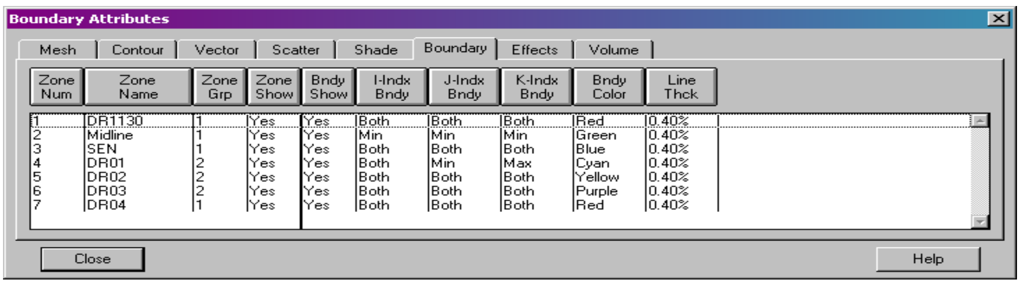



**Figure 10-4.** The Boundary Attributes dialog.

## 10.4. Specifying Which Boundaries are Displayed

For IJ-ordered zones, the available boundaries are the lines *I=1*, *I=IMax*, *J=1*, and *J=JMax*. When the Volume Attributes dialog’s Surfaces to Plot option is set to Boundary Cell Faces, Exposed Cell Faces, or Every Surface for IJK-ordered zones, the available boundaries are the boundaries of the surface areas, forming a “box” that contains the data. When the Volume Attributes dialog’s Surfaces to Plot option is set to one of the planes options, such as I-, J-, or K-planes, for IJK-ordered zones the boundaries are the boundaries of each plane (I-, J-, or K-plane). By default, all available boundaries are drawn when the Boundary layer is active. You can specify which of the available boundaries are plotted using either the Boundary Attributes dialog or the Quick Edit dialog.

To specify which boundaries to draw using the Boundary Attributes dialog:

1. From the Boundary Attributes dialog, select the zone or zones for which you want to specify which boundaries to draw.
2. Click I-Indx Bndy (or J-Indx Bndy or K-Indx Bndy). The options None, Min Only, Max Only, and Both appear.
3. Click on the desired option.

To specify which boundaries to draw using the Quick Edit dialog:

1. In the workspace, use the Selector tool to select the boundary of the zone for which you want to specify the boundaries to be drawn. When you select a zone boundary, an extra selection handle shows which of the zone's available boundaries is currently selected. Click on any other available boundary to select that boundary.
2. Call up the Quick Edit dialog from the sidebar.
3. Click on one of the following buttons in the Boundary display area:
   -  To display all available boundaries for the zone.
   -  To display the currently selected boundary.
   -  To turn off the currently selected boundary.
   -  To turn off all boundaries except the currently selected boundary.

# CHAPTER 11 Creating Contour Plots

Contour plots show the variation of one variable across the data field. Contour lines and contour flooding are examples of contour plots, as are multicolored mesh, scatter, and vector plots. In this chapter, however, we restrict our attention to plots having contour lines and/or contour flooding.

Plotting contours allows you to add an extra dimension to a plot, and thus contour plots may require one more variable than mesh plots of the same dimension. Where a 2-D mesh plot shows two variables, a contour plot can show three, and a 3-D contour plot can show four. This extra variable is called the contour variable.

The procedure for creating a contour plot is:

1. Read in a data set.
2. Select the Contour check box on the sidebar to activate the Contour zone layer. If no contour variable is currently assigned, the Contour Variable dialog appears with the default contour variable selected. You can either choose a different contour variable, or click Close to accept the default.
3. Deselect the Mesh check box on the sidebar. This turns off the Mesh zone layer, which is on by default.

The default contour plot is a flood contour plot, as shown in Figure 11-1, with 15 contour levels spanning a range calculated by Tecplot from the range of your contour variable.

## 11.1. Modifying Your Contour Plot

You can modify the attributes of your contour plot using either the Contour Attributes dialog or the Quick Edit dialog. You can control any of the following attributes from the Contour Attributes dialog, shown in Figure 11-2:

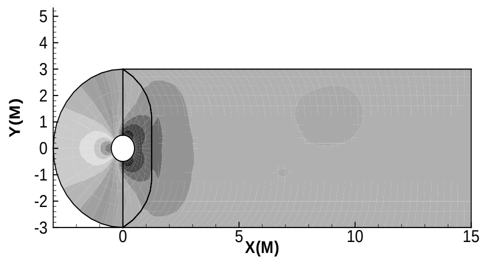



**Figure 11-1.** A contour plot of the cylinder data.

- Which zones are active. See Section 9.3.2, “Controlling Which Zones are Displayed.”
- Whether the contours are visible for each active zone. See Section 9.3.3, “Controlling Zone Layer Display.”
- The contour plot type. See Section 11.3, “Controlling the Contour Plot Type.”
- The contour line color. See Section 9.3.4, “Choosing Colors.”
- The contour line pattern. See Section 9.3.5, “Choosing a Line Pattern.”
- The contour line pattern length. See Section 9.3.6, “Choosing a Pattern Length.”
- The contour line thickness. See Section 9.3.7, “Choosing a Line Thickness.”
- Whether the lighting effect for contour flooding should be active for this zone. See Section 11.3.3, “Lighting Effects and Contour Flooding.”

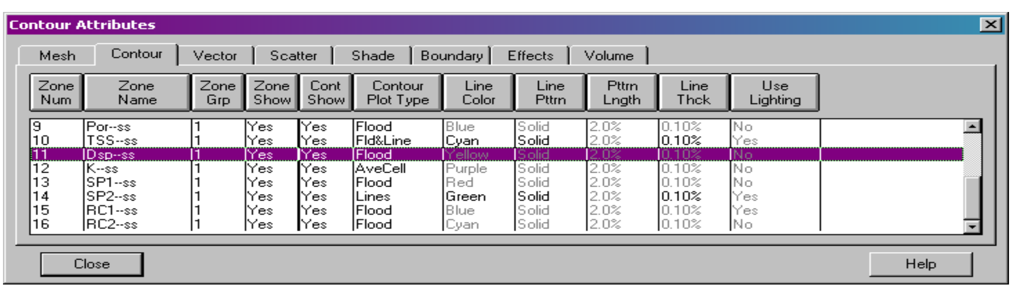



**Figure 11-2.** The Contour Attributes dialog.

The following attributes of contour plots are set on a frame-by-frame basis, rather than zone-by-zone:

- The contour variable. See Section 11.2, “Choosing a Contour Variable.”

- The contour line mode. See Section 11.3.1, “Controlling Contour Lines.”
- Contour levels. See Section 11.4, “Specifying Contour Levels.”
- The contour legend. See Section 11.7, “Creating a Contour Legend.”
- Contour labeling. See Section 11.8, “Contour Labels.”
- Adjustments to the color map, including the choice of Banded or Continuous color distribution. See Section 11.6, “Adjusting the Color Map for a Specific Frame.”

## 11.2. Choosing a Contour Variable

You choose a contour variable from the Contour Variable dialog. This dialog comes up automatically when you turn on the Contour zone layer for the first time in a frame, or can be accessed via the Contour Variable option on the Field menu. The Contour Variable dialog is shown in Figure 11-3. Simply choose one of the data set’s variables from the drop-down labeled Current Contour Variable.


**Figure 11-3.** The Contour Variable dialog.

## 11.3. Controlling the Contour Plot Type

Tecplot allows you to create contour plots of five different types:

- **Lines:** Draws lines of constant value of the specified contour variable.
- **Flood:** Floods regions between contour lines with colors from the global color map.
- **Both Lines and Flood:** Combines above two options.
- **Average Cell:** Floods cells or finite-elements with colors from the global color map according to the average value of the contour variable over the data points bounding the cell.
- **Corner Cell:** Floods cells or finite-elements with colors from the global color map according to the value of the contour variable at one corner of the cell.

By default, Tecplot uses the Flood type with Banded color distribution. Figure 11-4 shows examples of each of the contour plot types.

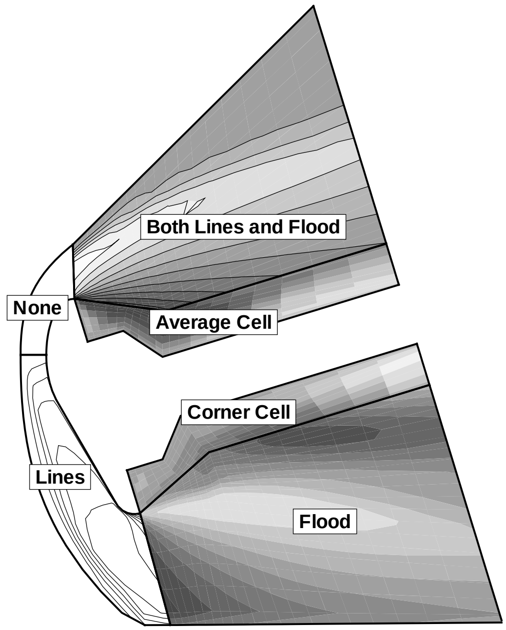



**Figure 11-4.** Contour plot types.

To modify the contour plot type:

1. From the Field menu, choose Contour Attributes. The Contour Attributes dialog appears.
2. On the dialog, select the zone or zones for which you want to modify the plot type.
3. Click Cont Plottype. A drop-down menu appears listing the available plot types.
4. Click on the desired plot type.

To modify the contour plot type from the Quick Edit dialog:

1. From the sidebar, click on Quick Edit. The Quick Edit dialog appears.
2. In the workspace, use the Selector tool to select the zone or zones for which you want to modify the plot type.
3. In the Quick Edit dialog, click on the appropriate button, as follows:

   -  Lines.
   -  Flood.

-  Both Lines and Flood.
-  Average Cell.
-  Corner Cell.

### 11.3.1. Controlling Contour Lines

You can control the color and thickness of the contour lines on a zone-by-zone basis, using the procedures of Section 9.3, "Modifying Your Field Plot." You can also specify the line pattern and pattern length on a zone-by-zone basis, but whether those settings are used depends on the current frame's contour line mode. The contour line mode determines how contour lines are drawn for all zones in the current frame's data set.

To specify the contour line mode:

1. From the Field menu, choose Contour Line Mode. The Contour Line Mode dialog appears, as shown in Figure 11-5.

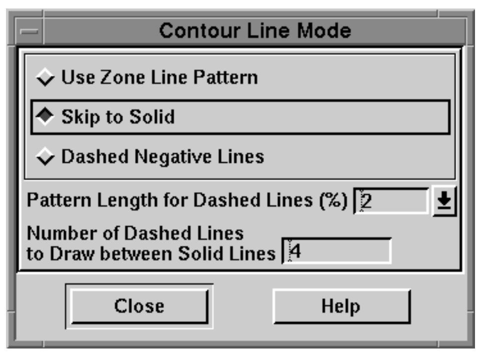



**Figure 11-5.** The Contour Line Mode dialog.

2. Choose one of the following options:
   - **Use Zone Line Pattern:** For each zone, draw the contour lines using the line pattern and pattern length specified in the Contour Attributes dialog.
   - **Skip to Solid:** Draw *n* dashed lines between each pair of solid lines, where *n* is an integer you enter in the text field Number of Dashed Lines to Draw between Solid Lines.
   - **Dashed Negative Lines:** Draw lines of positive contour variable value as solid lines and lines of negative contour variable value as dashed lines.

3. If you choose Skip to Solid or Dashed Negative Lines, specify a pattern length for the dashed lines.
4. If you choose Skip to Solid, enter the number of dashed lines to draw between solid lines.

### 11.3.2. Controlling Contour Flooding

A flooded contour plot is a contour plot in which the area between contour levels is filled (that is, flooded). Flooded contour plots give an immediate visual impression of how the contour variable is changing.

The distribution of colors used for contour flooding may be banded or continuous. When banded distribution is used for flooding a solid color is used between contour levels. If continuous color distribution is used the flood color will vary linearly in all directions. See Section 11.6.2, "Color Distribution Methods," for details on Tecplot's color distribution methods.

To create a flooded contour plot from the Contour Attributes dialog, set the contour plot type to either Flood or Both Lines and Flood. The area between adjacent contour levels is colored according the value of the contour variable, the color distribution method, the number of contour levels (banded distribution only), and the active color map. If you select the contour plot type Both Lines and Flood, the flooding is displayed along with the contour lines. (If the contour lines are multi-colored, you will not be able to see them against the contour flooding.)

In addition to the standard contour flooding, Tecplot supports two other types of flooded contour plots: average cell and corner. Unlike the standard flooding, which floods the regions between adjacent contour levels, these options flood individual cells or elements. The flooding is based on either the average value of the contour variable at the data points of the cell or element (for average cell) or the value at the primary corner of the cell or element (for corner flooding). The primary corner is either the first node of the cell (as specified in the data file's element definition), or the point of minimum I-, J-, and K-index.

When viewing a 3-D figure, either finite-element or IJK-ordered, each face of each element is colored individually. This is because each face may be part of two cells, each with a different value. For average cell contours, the flooding is based on the average value of the contour variable at the corners of the face. For corner contour flooding, the color is based on the value at the primary corner of the face.

### 11.3.3. Lighting Effects and Contour Flooding

Lighting effects may distort the surface color depending on the orientation of the surface with respect to the light source. You may want to turn off the lighting effects when creating flooded contour plots in 3-D to assure the surface colors match those in the contour legend. The easiest way to do this for zones is via the Lighting Zone Effects check box on the sidebar.

To maintain shading on objects with a combination of plain surface shaded zones and flooded contour zones make sure the sidebar's Lighting Zone Effects check box is selected. You must also turn off lighting zone effects on a zone-by-zone basis with the Use Lighting option on the Contour page of the Plot Attributes dialog.

## 11.4. Specifying Contour Levels

A contour level is a value at which contour lines are drawn. The number of contour levels is the number of contour lines that will be drawn. The range of contour levels is the interval between the minimum contour level and the maximum contour level.

You control contour levels using the Contour Levels dialog, shown in Figure 11-6. Call up this dialog by choosing Contour Levels option from the Field menu. From this dialog, you can perform any of the following tasks:

- Specify a new range or number of contour levels.
- Add levels to an existing set of contour levels.
- Remove selected contour levels.

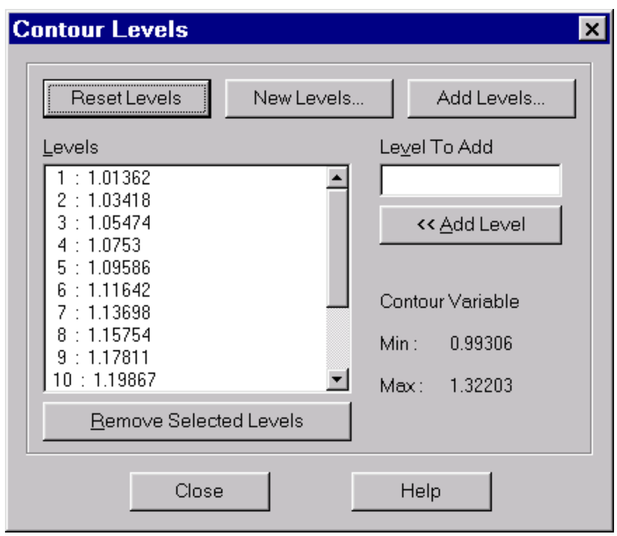



**Figure 11-6.** The Contour Levels dialog.

### 11.4.1. Specifying the Range or Number of Contour Levels

The number and range of contour levels should usually be modified together.

To modify the number of levels while letting Tecplot determine the range, click Reset Levels on the Contour Levels dialog. An Enter Value dialog appears with the default (or current) number of contour levels, as shown in Figure 11-7. Enter the desired number of contour levels and click OK.,

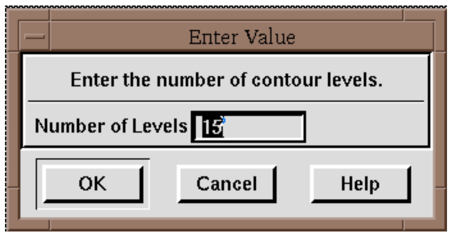



**Figure 11-7.** Entering the number of contour levels.

To modify the number of levels and control the range yourself, click New Levels on the Contour Levels dialog. The Enter Contour Level Range dialog appears with the current settings. The dialog is shown in Figure 11-8.

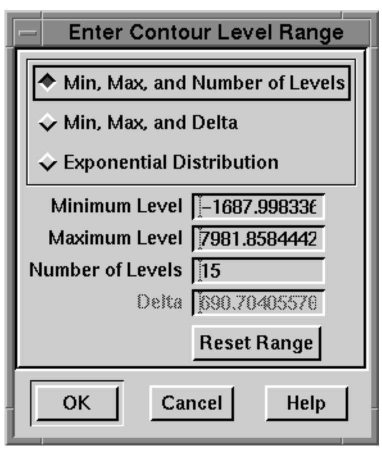



**Figure 11-8.** The Enter Contour Level Range dialog.

You can specify the range and number of levels in any of three ways:

- As a minimum and maximum level value, together with the number of levels to be distributed equally through the range. Choose this option (the default) by selecting the check box labeled Min, Max, and Number of Levels, then filling in the appropriate text fields.
- As a minimum and maximum level value, together with a delta, that is, the difference or change in contour value between two adjacent contour levels. You choose this option by selecting the check box labeled Min, Max, and Delta, then filling in the appropriate text fields. Tecplot generates contour levels beginning at the minimum level specified, then at intervals of Delta, until adding Delta would result in a level higher than the maximum level specified. Thus the highest actual contour level is usually somewhat less than the maximum level unless Delta is chosen precisely.
- As a minimum and maximum level value, together with the number of levels to be distributed exponentially through the range. You choose this option by selecting the check box labeled Exponential Distribution, then filling in the appropriate text fields.

In specifying a range, you can refer to the main Contour Levels dialog, which displays the minimum and maximum range of the contour variable. You can specify contour levels that extend beyond the range of the contour variable; any levels outside the range of the data are not displayed.

### 11.4.2. Adding Contour Levels

You can add contour levels to an existing set of contours. You might do this to get a clearer idea of how the contour variable is varying in areas with few visible contour lines.

You can add new levels in any of three ways:

- Add a new range of contour levels to the existing set by clicking the Add Levels option in the Contour Levels dialog and then using the Enter Contour Level Range dialog as described in Section 11.4.1, “Specifying the Range or Number of Contour Levels.”
- Enter a value in the Level To Add text field in the Contour Levels dialog and then clicking Add Level.
- Choose  from the sidebar, then click at any location in the contour plot where you would like a new contour level. Tecplot adds a new contour level that goes through the specified point. By holding down the mouse button you can drag and interactively position the new contour level until you release the button. While using this tool you may also press the “-” key to switch to the Remove Contour tool.

### 11.4.3. Removing Contour Levels

You can remove contour levels in either of two ways:

- In the Levels scrolled list in the Contour Levels dialog, select one or more contour levels, then click Remove Selected Levels.

- Choose  from the sidebar, then click on any contour line in your contour plot. Tecplot deletes the specified contour level, or the nearest contour level to the specified point. While using this tool you may also press the "-" key to switch to the Add Contour tool.

### 11.4.4. Adjusting Contour Levels

You can interactively adjust a contour level with the  tool. Select the tool from the sidebar. Hold down the Ctrl key; then click and drag the contour level you want to adjust. Move the contour to the desired location and release the mouse button. The value of the contour level will change as can be viewed in the Contour Levels dialog.

## 11.5. Controlling the Global Color Map

The colors used in flooded contour plots are determined by the global color map, controlled in the Workspace menu, and by frame-specific color options controlled in the Field menu. This section discusses the global color map; frame-specific color options are discussed in Section 11.6, "Adjusting the Color Map for a Specific Frame."

By default, Tecplot uses a color map called Small Rainbow, which is a rainbow of colors from blue to cyan to green to yellow to red. This default color map is called Small Rainbow to distinguish it from another color map, Large Rainbow, which adds purple and white beyond the red. The color map is used by all frames; if you change the color map to modify the look of one frame, all frames with contour flooding or any form of multi-colored will be modified as well.

To choose a color map:

1. From the Workspace menu, choose Color Map. The ColorMap dialog appears, as shown in Figure 11-9.
2. From the Base Color Map drop-down menu, choose one of the following color maps:
   - **Small Rainbow:** Five color spectrum from blue to cyan to green to yellow to red.
   - **Large Rainbow:** Seven color spectrum from blue to cyan to green to yellow to red to purple to white.
   - **Modern:** Seven color spectrum; within each color band colors change in intensity from dark to light.
   - **GrayScale:** Color spectrum from black to white.
   - **Wild:** random Color spectrum. Wild is different each time you select it.

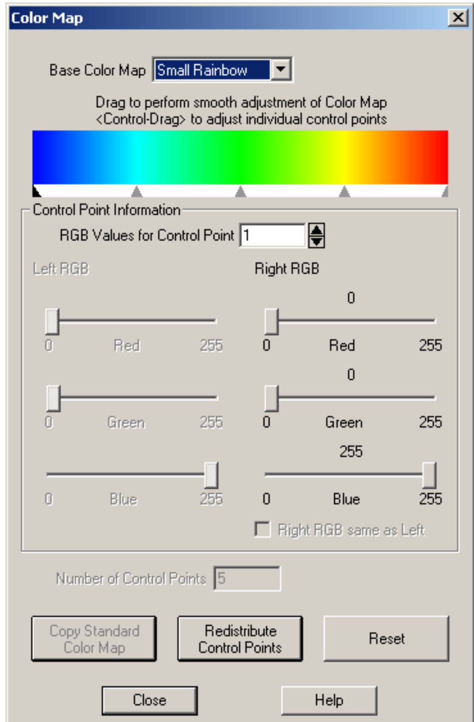



**Figure 11-9.** The Color Map dialog.

- **Two Color:** A two-color spectrum.
- **User-Defined:** A customizable version of one of the first four options above. You can add or delete control points, as well as change RGB values for each control point. See Section 11.5.2, “Modifying a User-Defined Color Map,” for details on modifying a User Defined color map.
- **Raw User-Defined:** A customizable version of one of the first four options above. To customize the color map, however, you must save your Raw User-Defined map to a file using the Copy Color Map to File option in the Workspace menu. See Section 11.5.3, “Modifying a Raw User-Defined Color Map,” for details on modifying a Raw User Defined color map.

You can modify any color map, except the Raw User-Defined color map, using the controls in the Color Map dialog.

### 11.5.1. Modifying a Standard Color Map

You can modify the standard color maps either by altering the position of the color map control points or by modifying the RGB values of the control points. Altering the position of the control points allows you to alter the proportions of colors in the spectrum. Modifying the RGB values of the control points changes the spectrum itself. All of the control points except the first and last are two-sided; they have both a left- and a right-RGB value. This allows you to define color maps such as Modern, which has sharp demarcations between color bands. To define smoothly varying color maps, select the Right RGB Same as Left check box for each two-sided control point. Except for the Wild and Modern color maps (and, if suitably modified, the Raw User-Defined color map), all color maps vary smoothly between control points.

To alter the position of control points:

1. Move the pointer into the color bar in the Color Map dialog.
2. Drag a point on the color bar. The entire color bar, including the control points, is distorted relative to the point where the drag started. If the point is dragged to the right, control points to the left of the drag point become more widely spaced, while control points to the right of the drag point are moved closer together. An XORed line shows the point being dragged. To move just a single control point, which does not affect the position of other control points, Ctrl-drag on the color bar to select the nearest control point, then drag as before. To return the control points to their original positions, click Redistribute Control Points.

To modify the RGB values for a control point:

1. Select the control point for which you want to modify RGB values. You can do this either by entering a value (or using the up and down arrows to choose a value) in the field labeled RGB Values for Control Point, or by Ctrl-clicking on the control point in the color bar.
2. Use the three sliders under the heading Left RGB to specify the left-RGB colors for the control point. (These sliders are disabled for the left-most control point.)
3. Use the three sliders under the heading Right RGB to specify the right-RGB colors for the control point, or select the check box Right RGB Same as Left to use the same values you just set for the left-RGB colors. (The Right RGB sliders are disabled for the right-most control point, or if the check box Right RGB Same as Left is selected.)

To reset the RGB values to their original values (and also reposition the control points in their original locations), click Reset.

### 11.5.2. Modifying a User-Defined Color Map

You can modify a User-Defined color map in the same way you modify a standard color map, by altering the position of control points or modifying the RGB values of the control points, but you have the added ability to modify the number of control points.

To change the number of control points in a user defined color map:

1. Choose the User Defined color map. By default, the User-Defined color map has the same settings as the standard Small Rainbow color map.
2. Enter a value from two through nine in the Number of Control Points field.

If you choose a number greater than the current value, the new control points are added to the right of the existing control points; click Redistribute Control Points to see all your control points. If you choose a number less than the current value, the control points are removed right-most first.

### 11.5.3. Modifying a Raw User-Defined Color Map

You can modify a Raw User-Defined color map only by saving it to a file and then editing the resulting file, which consists of RGB triplets for every color in the spectrum. You can modify these RGB triplets as you want, using any ASCII text editor. In most cases, you want to use the User-Defined color map rather than the Raw User-Defined, since you cannot edit the raw color map in Tecplot.

### 11.5.4. Color Map Files

The position of color map control points and their RGB values can be stored in color map files; you can then edit the color map files to modify either the position or RGB values of the control points.

To create a color map file:

1. From the menu, choose Copy Color Map to File. The Write Color Map dialog appears.
2. Specify a file name for the color map.
3. Click OK.

To use the saved color map in a new plot, choose Paste Color Map from File on the workspace menu. The color map file is a Tecplot macro file with a limited set of commands (only **`$!COLORMAP`** and **`$!COLORMAPCONTROL`** commands are allowed). The first part of the color map file generated from the Large Rainbow color map is shown below:

```
#!MC 900
$!COLORMAP
```

```
 CONTOURCOLORMAP = LGRAINBOW
$!COLORMAPCONTROL RESETTOFACTORY
$!COLORMAP
 LGRAINBOW
    {
    CONTROLPOINT 1
      {
      COLORMAPFRACTION = 0
      LEADRGB
        {
        R = 0
        G = 0
        B = 255
        }



      TRAILRGB
        {
        R = 0
        G = 0
        B = 255
        }
      }
. . .
```

## 11.6. Adjusting the Color Map for a Specific Frame

Although the color map is global, affecting all frames, there are some adjustments you can make that apply only to the current frame. These adjustments allow you to customize the look of a single contour plot without changing the global color map. Adjustments to the color map for individual frames are made from the Contour Coloring Options dialog, shown in Figure 11-10.

Here are the color properties you can change for each frame:

- The color distribution method may be continuous or banded.
- The color map may be cut off such that coloring outside of a given range is not done at all.
- The color map can be reversed.
- The color map can contain multiple cycles.

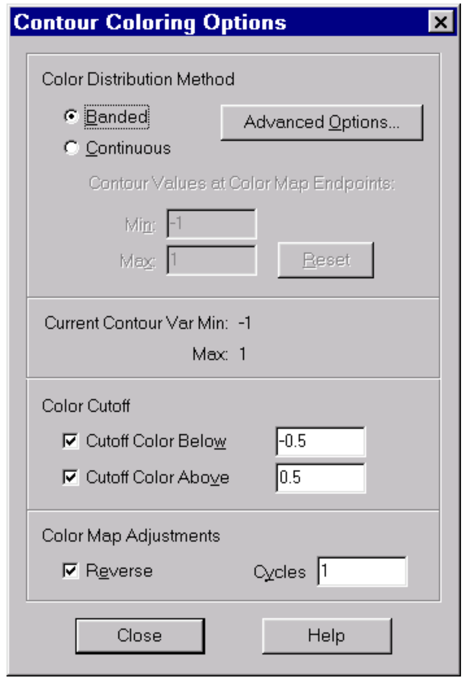



**Figure 11-10.** The Contour Coloring Options dialog.

## 11.6.1. Color Distribution Methods

The color distribution method determines how to look up the colors in the global color map. There are two options; Banded and Continuous.

**11.6.1.1. Continuous Color Distribution.** Continuous color distribution assigns linearly varying colors to all multi-colored objects or contour flooded regions. Continuous coloring is currently only available for 3D frame mode.

**11.6.1.2. Banded Color Distribution.** Banded color distribution is closely tied to the contour levels. A solid color is assigned for all values within the band between two levels. For contour flooding this means the area between two levels is filled with a single solid color. For multi-color scatter symbols or vectors this means all scatter symbols or vectors that have a contouring value between two contour levels will be assigned the same color.

When banded coloring is in effect the following options are available:

- Color bands can be zebra shaded. This in effect colors every other band with a specific color (or no color at all).
- Specific contour bands can be assigned a unique basic color. This is useful for forcing a particular region to use blue, for example, to designate an area of water. You can define up to 16 color overrides.

For these selections, click Advanced Options on the Contour Coloring Options dialog. The Advanced Band Coloring Options dialog appears, as shown in Figure 11-11.

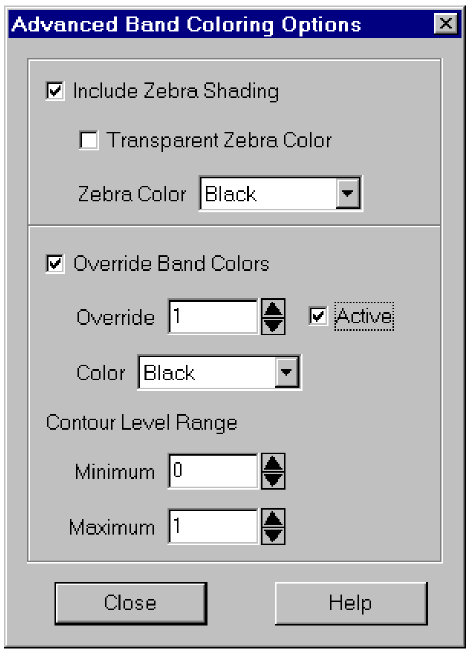



**Figure 11-11.** The Advanced Band Coloring Options dialog.

## 11.6.2. Color Cutoff

Color cutoff lets you specify a range within which contour flooding and multi-colored objects, such as scatter symbols, are displayed. For example, you may specify that only contour flooding in the range of -4.5 to 4.5 should be displayed; contour flooding outside this range is not plotted.

To use color cutoff:

1. From the Field menu's Contour options choose Contour Coloring Options.
2. To set a minimum color cutoff select the Cutoff Color Below check box and enter a value in the text field.
3. To set a maximum color cutoff select the Cutoff Color Above check box and enter a value in the text field.

A flooded contour plot before and after color cutoff has been applied is shown in Figure 11-12.

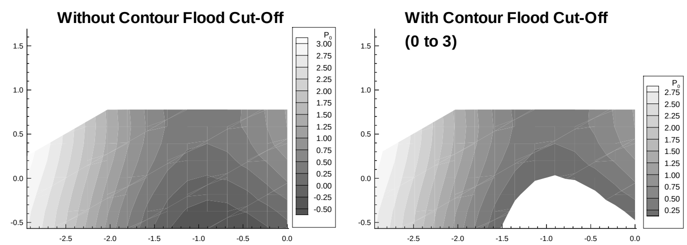



**Figure 11-12.** Flooded contour plot without and with flood cutoff.

### 11.6.3. Reversing the Color Map

You can reverse the color map by selecting the check box at the bottom of the Contour Coloring Options dialog. Two plots, one with the color map going in the default direction, and one with the color map reversed, are shown in Figure 11-13.

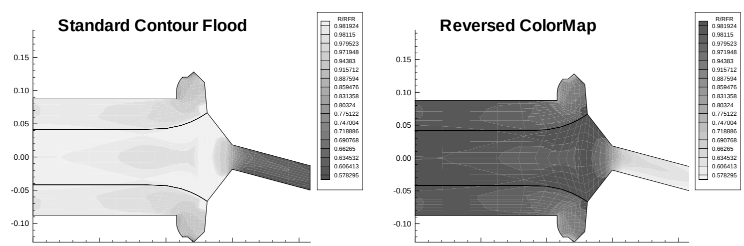



**Figure 11-13.** Flooded contour plot with a standard and a reversed color map.

### 11.6.4. Color Map Cycles

You may choose to cycle the color map. This is useful if you have data where there is a great deal of activity in multiple ranges of the contour variable and you want to cycle through all colors in each region. A plot with the color map cycled two times is shown in Figure 11-14.

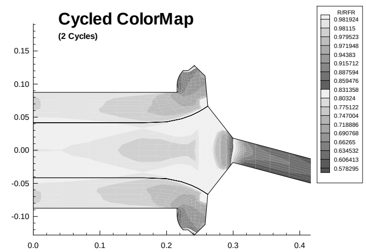



**Figure 11-14.** Flooded contour plot with the color map cycled two times.

## 11.7. Creating a Contour Legend

A contour legend is a key to the contour flooding in your flooded contour plot. It relates the displayed colors to the actual values of the contour variable. The contour legend can also relate contour level numbers to contour values. This is useful if you are using contour number labels. To create a contour legend, do the following:

1. From the Field menu, select Contour Legend. The Contour Legend dialog appears.
2. Select the check box labeled Show Contour Legend.
3. Select the check box labeled Show Header to include a legend heading that includes the name of the contour variable.
4. If you want to include black lines between the color bands representing each level, select the check box labeled Separate Color Bands.
5. Choose an orientation for the legend by selecting one of the option buttons Align Vertical or Align Horizontal.
6. Specify the location of the upper left corner of the legend by entering values in the X (%) and Y (%) text fields. Enter X as a percentage of the frame width and Y as a percentage of the frame height. (The legend is also moveable interactively.)
7. Enter the number of levels between each entry in the legend in the Level Skip text field, and enter the line spacing between entries in the Line Spacing text field. Together, these two text fields control the overall size of your legend.
8. Format the text for the legend by choosing a color and font, and specifying the text height as a percentage of the frame height. Enter the desired line spacing in the Line Spacing text field.

9. Specify the format for numeric values in the legend. The available options are the same as for axis tick mark labels; see Section 17.5.3, “Tick Mark Label Formats.”
10. Select which kind of box you want drawn around the legend (No Box, Filled, or Plain). If you choose Filled or Plain, format the box using the following controls:
    - **Line Thickness:** Specify the line thickness as a percentage of frame height.
    - **Box Color:** Choose a color for the legend box outline.
    - **Fill Color (Filled only):** Choose a color for the legend box fill.
    - **Margin:** Specify the margin between the legend text and legend box as a percentage of the text height.

## 11.8. Contour Labels

Contour labels are labels that identify particular contour levels either by number or by value. You can place contour labels interactively, or have Tecplot create them for you automatically. You can also have Tecplot create a set of contour labels automatically, then interactively add contour labels to this saved set. You control contour labels with the Contour Labels dialog under the Field menu, and with the Add Contour Label mouse mode tool from the sidebar.

To add contour labels to your plot, you can use either of the following procedures. The first describes using the Add Contour Label mode tool; the second describes using the Contour Labels dialog to have Tecplot automatically generate the contour labels. If you are using the Add Contour Label tool, you still use the Contour Labels dialog to specify whether contour level numbers or values are used and to specify formatting and alignment options for the labels.

Contour labels show the value or number of the nearest contour level. In other words, if you add a label between two contour lines, the label will show the value or number of the nearest line. This can be misleading when contour labels are far away from contour lines.

To add contour labels interactively using the Add Contour Label mode:

1. Create a plot with an active Contour zone layer.
2. From the sidebar, choose the Add Contour Label mode by clicking .
3. In the workspace, click on the location at which you want a contour label to appear. By default, Tecplot uses contour values and aligns the labels with the contour line. You can modify the following options using the Contour Labels dialog.
    - The label color.
    - The label font.
    - The label size.

- The numeric format. The available formats are described in Section 17.5.3, “Tick Mark Label Formats.”
- Whether there is a color fill behind the labels, and if so, its color and the margin of fill around the labels.
- Label alignment. If the Align Next User-Positioned Label is selected, the next label placed is aligned with the contour line. Otherwise, the label is written with normal, upright text.

To have Tecplot generate contour labels automatically with each redraw:

1. Create a plot with an active Contour zone layer.
2. From the Field menu, choose Contour Labels. The Contour Labels dialog appears as shown in Figure 11-15.

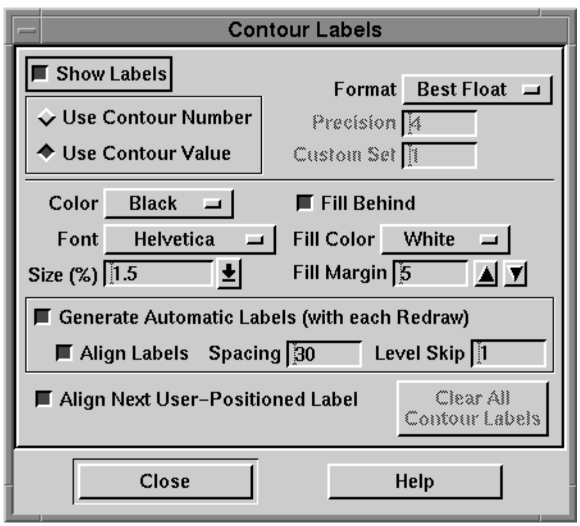



**Figure 11-15.** The Contour Labels dialog.

3. Select the check box labeled Show Labels.
4. Choose whether contour numbers or contour values are used in the labels by selecting the appropriate option.
5. If you want the labels aligned with the contour lines, select the Align Labels check box.
6. Specify the spacing between labels in frame units.
7. Specify the number of levels to skip in the Level Skip text field. The default of 1 labels all contour labels.

8. Select the Generate Automatic Labels (with each Redraw) check box.

At each Redraw, Tecplot creates a new set of contour labels. At any time, you can deselect the Generate Automatic Labels (with each Redraw) check box, and Tecplot will retain the last set of labels generated. Thus you can create labels for several redraws, then save a set to which you can interactively add more labels.

When Generate Automatic Labels is deselected, you can click Clear All Contour Labels to erase the current set of contour labels.

# CHAPTER 12 Creating Vector Plots

Vector plots are field plots of the direction and or magnitude of vector quantities. The vector quantities can be displacements, velocities, forces, or anything else that can be represented by vectors. You create vector plots by activating the Vector layer in the Tecplot sidebar, and, if you have not done so already, specifying two or three vector component variables.

One important use of vector components in Tecplot is to allow you to compute the trajectories of massless particles in a steady-state velocity field. These trajectories are called streamtraces. Streamtraces do not need to be created in vector plots, even though they require vector components. For this reason, they are discussed in a separate chapter. See Chapter 13, "Streamtraces."

## 12.1. Creating a Vector Plot

When you select the Vector check box in the Tecplot sidebar, Tecplot checks to see whether vector components have been assigned for the current data set in the current frame mode, whether 2D or 3D. If you have not assigned vector components, the Select Variables dialog appears. In 2D frame mode, the Select Variables dialog allows you to choose two vector components, as shown in Figure 12-1.

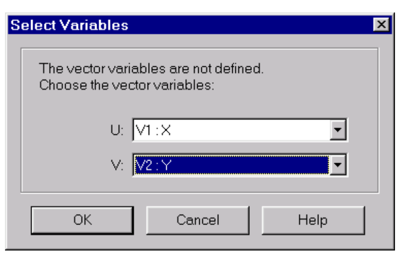



**Figure 12-1.** The Select Variables dialog for assigning 2-D vector components.

Choose variables by selecting the desired U-, V-, and in 3D frame mode, W-variables from their respective drop-downs. You may select any of the current data set's variables as any component. You can change the component variables at any time by choosing Vector Variables from the Vector sub-menu of the Field menu.

Once you have selected the Vector check box and have chosen your vector components your vector plot will appear as shown in Figure 12-2 for the cylinder data. If vectors are not visible, see 12.5, "Controlling Vector Length."

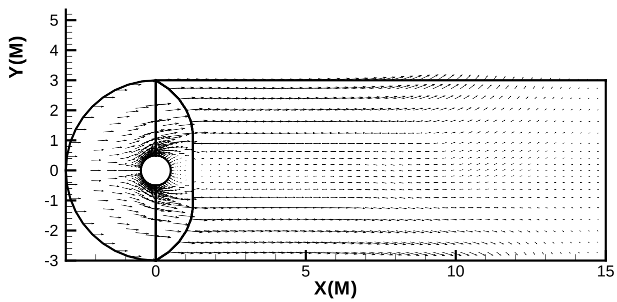



**Figure 12-2.** A vector plot of the cylinder data.

## 12.2. Modifying Your Vector Plot

You can modify your vector plot attributes using either the Vector Attributes dialog or the Quick Edit dialog. You can control any of the following attributes from the Vector Attributes dialog (Figure 12-3):

- Which zones are active. See Section 9.3.2, "Controlling Which Zones are Displayed."
- Whether the vectors are visible for each active zone. See Section 9.3.3, "Controlling Zone Layer Display."
- The vector plot type. See Section 12.3, "Controlling the Vector Plot Type."
- The arrowhead style. See Section 12.4.1, "Controlling Arrowhead Style."
- Whether 3-D vectors are tangent vectors or regular vectors. See Section 12.7.1, "Tangent Vectors."
- The vector color. See Section 9.3.4, "Choosing Colors."
- The vector line pattern. See Section 9.3.5, "Choosing a Line Pattern."
- The vector line pattern length. See Section 9.3.6, "Choosing a Pattern Length."
- The vector line thickness. See Section 9.3.7, "Choosing a Line Thickness."
- The vector spacing. See Section 12.6, "Controlling Vector Spacing."

The following attributes are assigned on a frame-by-frame basis, rather than zone-by-zone:

- Vector lengths. See Section 12.5, "Controlling Vector Length."
- Arrowhead angle and size. See Section 12.4, "Controlling Vector Arrowheads."
- The optional reference vector. See Section 12.8, "Displaying a Reference Vector."

## 12.3. Controlling the Vector Plot Type

Tecplot allows you to plot vectors of four different types:

- **Tail at point:** Vectors are drawn with the tail of the vector positioned at the data point, which for ordered data is a corner of the cell.
- **Head at point:** Vectors are drawn with the head of the vector positioned at the data point.
- **Anchor at midpoint:** Vectors are drawn with the midpoint of the vector positioned at the data point.
- **Head only:** A vector arrowhead (but no tail) is drawn with the head of the arrow centered at the data point.

By default, Tecplot uses the "Tail at Point" type. Figure 12-4 shows examples of each of the vector plot types.

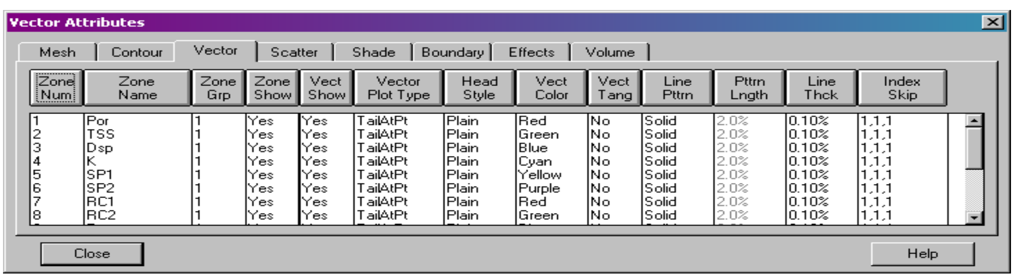



**Figure 12-3.** The Vector Attributes dialog.

To modify the vector plot type:

1. Choose Vector Attributes from the Vector sub-menu of the Field menu. The Vector Attributes dialog appears as shown in Figure 12-3.
2. In the Vector Attributes dialog, select the zone or zones for which you want to modify the plot type.
3. Click Vect Plottype. A drop-down appears listing the available plot types.

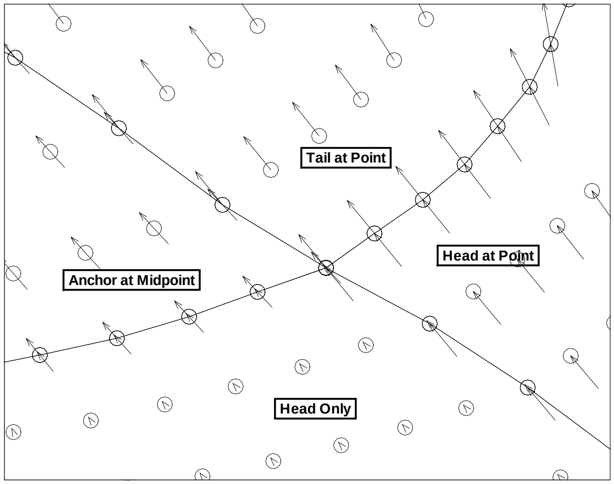



**Figure 12-4.** Vector plot types.

4. Click on the desired plot type.

To modify the vector plot type from the Quick Edit dialog:

1. On the sidebar, click Quick Edit. The Quick Edit dialog appears.
2. In the workspace, use the Selector tool to select the zone or zones for which you want to modify the plot type.
3. On the Quick Edit dialog, click on the appropriate button, as follows:
   - Tail at point.
   - Head at point.
   - Anchor at midpoint.
   - Head only.

## 12.4. Controlling Vector Arrowheads

Tecplot allows you a good deal of control over your vector arrowheads. You can control the style of the arrowhead, its size, and the angle it makes with the vector. The style of the arrowhead can change from zone to zone; the size and angle are global attributes affecting vectors in all zones.This section explains how to perform these tasks.

### 12.4.1. Controlling Arrowhead Style

You can assign arrowhead styles on a zone-by-zone basis. Tecplot arrowheads come in three styles:

- **Plain:** Line segments drawn from the head of the vector.
- **Filled:** Filled isosceles triangles with apex at the head of the vector.
- **Hollow:** Hollow isosceles triangles with apex at the head of the vector.

By default, Tecplot draws plain arrowheads. Figure 12-5 shows an example with all three arrowhead styles.

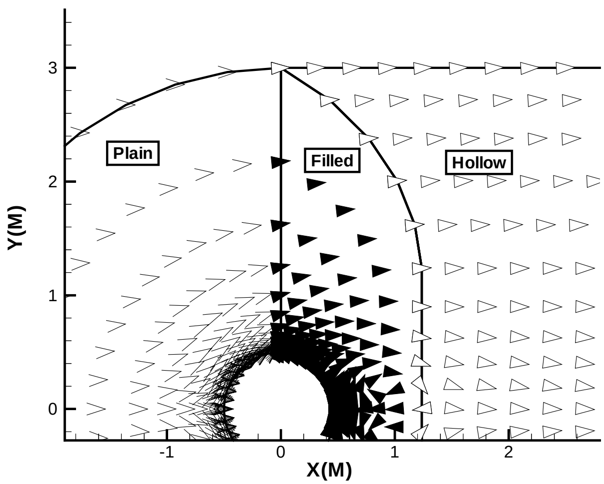



**Figure 12-5.** Examples of Tecplot's three arrowhead styles.

To modify the arrowhead style for a zone or zones:

1. Choose Vector Attributes from the Vector sub-menu of the Field menu. The Vector Attributes dialog appears.
2. Select the zone or zones for which you want to modify the arrowhead style.
3. Click Head Style. A drop-down appears containing the available choices.
4. Click on the desired arrowhead style.

or

1. In the workspace, use the Selector tool to select the zone or zones for which you want to modify the arrowhead style.

2. On the sidebar, click Quick Edit. The Quick Edit dialog appears.
3. In the Quick Edit dialog, click on the button for the desired arrowhead style, as follows:
   - Plain arrowhead style.
   - Filled arrowhead style.
   - Hollow arrowhead style.

## 12.4.2. Controlling Arrowhead Size

You can specify arrowhead sizes as either a fraction of the vector length or in frame units (that is, as a percentage of the frame height). Arrowhead size is a global attribute; it applies to all arrowheads in all zones in the current frame. By default, Tecplot specifies size as a fraction of the vector length.

To modify the arrowhead size:

1. Choose Vector Arrowheads from the Vector sub-menu of the Field menu. The Vector Arrowheads dialog appears as shown in Figure 12-6.

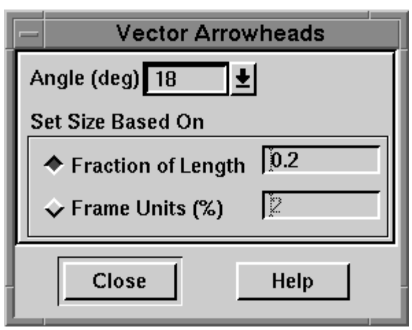



**Figure 12-6.** The Vector Arrowheads dialog.

2. Select one of the two option buttons in the box labeled Set Size Based On: either Fraction of Length or Frame Units (%).
3. Enter a value in the text field to the right of the selected option button. Use a fraction (a decimal value from zero to one) in the Fraction of Length text field. Use a percentage (a decimal value from zero to 100) in the Frame Units (%) field. Entering a value of zero in either field effectively “turns off” the vector arrowhead in the current frame.

### 12.4.3. Controlling Arrowhead Angle

Arrowhead angle is another global attribute; you assign one angle for all vector arrowheads in all zones in the current frame. The arrowhead angle is the angle (in degrees) that one side of the arrowhead makes with the vector; thus, the apex angle is twice the arrowhead angle. For example, to create hollow equilateral triangles as arrowheads, specify an arrowhead angle of 30 globally in conjunction with an arrowhead style of hollow for all zones.

To specify the arrowhead angle:

1. Choose Vector Arrowheads from the Vector sub-menu of the Field menu. The Vector Arrowheads dialog appears.
2. In the field labeled Angle (deg), either enter a value from 1 to 90, or choose a value from the drop-down, indicated by the down-arrow button.

## 12.5. Controlling Vector Length

You can specify the length of vectors in any of three different ways. In the first two, the length of any given vector is proportional to the vector magnitude, while in the third, all vectors have the same length. The difference between the first two methods is the units used to specify the relative size of the vectors, either grid units or screen centimeters. Vector length is a global attribute — it applies to all zones in the current frame.

To specify the vector length:

1. Choose Vector Length from the Vector sub-menu of the Field menu. The Vector Length dialog appears as shown in Figure 12-7.

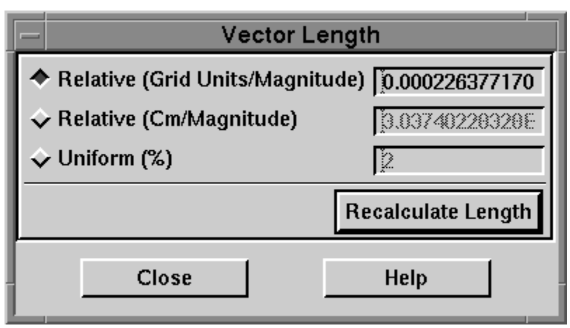



**Figure 12-7.** The Vector Length dialog.

2. Select one of the three option buttons:

- **Relative (Grid Units/Magnitude):** specify the vector length as the number of grid units per unit of vector magnitude.
- **Relative (Cm/Magnitude):** specify the vector length as the number of centimeters per unit of vector magnitude.
- **Uniform (%):** specify the vector length as a percentage of frame height.

3. Enter a value in the text field to the right of the selected option button. For either of the “Relative” options, the value you specify is a scale factor which is multiplied by the vector magnitude to determine the length of the vector.

By default, Tecplot calculates a reasonable default based on the size of the longest vector. You can have Tecplot recalculate this default length by clicking Recalculate Length.

## 12.6. Controlling Vector Spacing

You can draw every vector or specify a skip factor that lets you plot only every *n*th vector (in I-, J-, or K- coordinates). Spacing the drawn vectors is useful in situations where you have an extensive vector field, and plotting all the vectors makes the general flow of the vector field difficult to discern. For example, Figure 12-8 shows the cylinder data with every other vector shown in the I direction and every third vector shown in the J direction. Compare this to the vector plot shown in Figure 12-2.

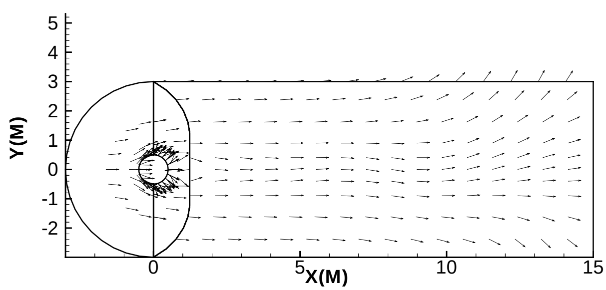



**Figure 12-8.** A vector plot with Index Skip specified.

To specify the vector spacing:

1. Choose Vector Attributes from the Vector sub-menu of the Field menu. The Vector Attributes dialog appears.
2. Select the zone or zones for which you want to specify the vector spacing.
3. Click Index Skip. A drop-down appears with the options No Skip and Enter Skip. (No Skip is the default.)

4. Click on the desired option. If you click on Enter Skip, the Enter Index Skipping dialog appears, as shown in Figure 12-9 below.
5. (Enter Skip only) Enter the desired values of I-Skip, J-Skip, and K-Skip. Figure 12-8 was created with the following settings: *I-Skip=2, J-Skip=3, K-Skip=1*..

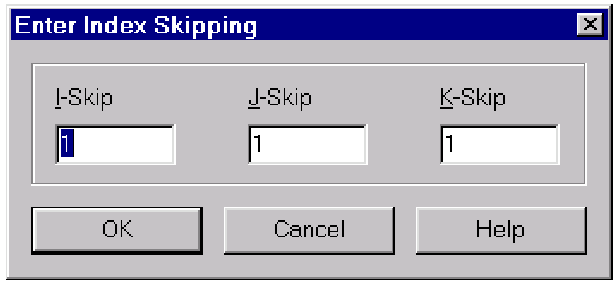



**Figure 12-9.** The Enter Index Skipping dialog.

For irregular and finite-element data, only the I-Skip has an effect, skipping through nodes in the order they are listed in the data file.

# 12.7. Creating 3-D Vector Plots

To create a 3-D vector plot, simply read a data set into Tecplot, choose the 3D frame mode, and select the Vector check box in the Tecplot sidebar. Three-dimensional vector plots require three vector components, U, V, and W, for the three axes X, Y, and Z. The Select Variables dialog appears asking you to specify vector variables.

## 12.7.1. Tangent Vectors

In 3D frame mode, Tecplot allows you to display 3-D surface tangent vectors. These are 3-D vectors that have been projected onto the 3-D surface; in other words, the component of the 3-D vectors normal to the surface is removed, leaving only the component parallel to the surface. Figure 12-10 shows how tangent vectors compare to regular vectors.

To select tangent vectors for a zone:

1. From the Vector sub-menu of the Field menu, choose Vector Attributes. The Vector Attributes dialog appears.
2. Select the zone or zones for which you want to draw tangent vectors.
3. Click Vect Tang. A drop-down appears with the choices Yes and No.
4. Click Yes.

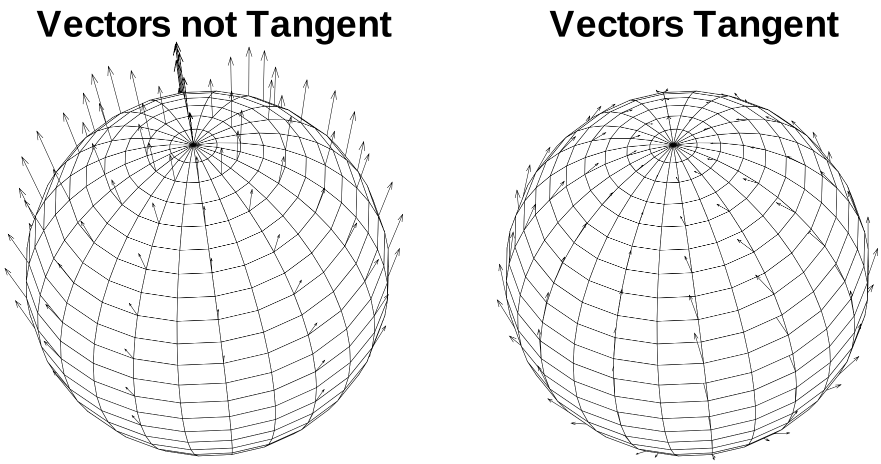



**Figure 12-10.** Regular vectors compared to tangent vectors.

Tangent vectors are drawn on 3-D surfaces only where it is possible to determine a vector normal to the surface. A plot where multiple surfaces intersect each other using common nodes is a case where tangent vectors are not drawn because there is more than one normal to choose from. An example of this would be a volume IJK-ordered zone where both the I- and J-planes are plotted. If tangent vectors cannot be drawn then regular vectors are plotted instead.

### 12.7.2. Lengths of 3-D Vectors

Since 3-D vectors are plotted in the plane of the screen, a 3-D vector's length will depend on both the vector length settings and the orientation of the vector. The length may be distorted even further if the vector length setting is Relative and the 3-D projection is Perspective.

## 12.8. Displaying a Reference Vector

A reference vector is a vector of specified magnitude placed on the plot as a measure against all other vectors. You can specify whether to show a reference vector, and if so, its color, orientation, line thickness, magnitude, and position.

To display a reference vector:

1. From the Vector sub-menu of the Field menu, choose Reference Vector. The Reference Vector dialog appears as shown in Figure 12-11.
2. Select the Show Reference Vector check box.

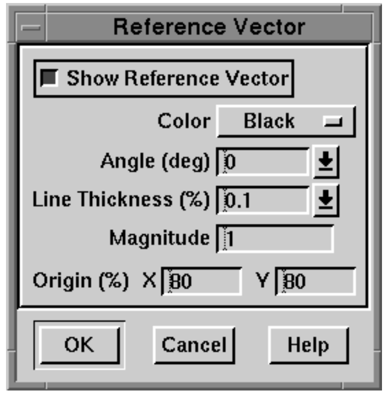



**Figure 12-11.** The Reference Vector dialog.

3. Modify any of the following options, as desired:
   - **Color**: Choose a color from the drop-down of Tecplot's basic colors.
   - **Angle (deg):** Enter the orientation of the vector in degrees from horizontal, or choose a value from the drop-down.
   - **Line Thickness (%):** Enter the desired line thickness or choose a value from the drop-down.
   - **Magnitude:** Enter the magnitude of the reference vector. The units correspond to those of the vector components.
   - **Origin (%):** Enter the coordinates of the starting point of the reference vector, as a percentage of the frame width (X-coordinate) and frame height (Y-coordinate).
4. Click OK to close the dialog.
5. On the sidebar, click Redraw.

Figure 12-12 shows a plot with a reference vector.

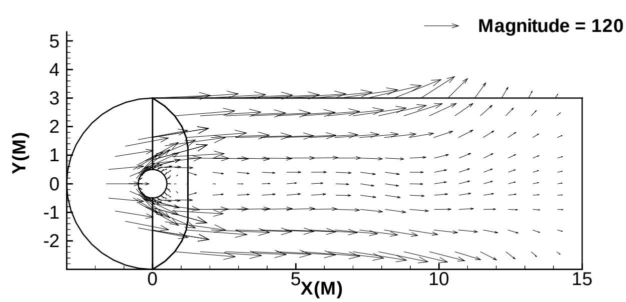



**Figure 12-12.** Vector plot with reference vector.

# CHAPTER 13 Streamtraces

A streamtrace is the path traced by a massless particle placed at an arbitrary location in a steady-state vector field. Tecplot's streamtrace features may be used to illustrate the nature of the vector field flow in a particular region of the plot.

Because streamtraces are dependent upon a vector field, you must define vector components before creating streamtraces in Tecplot. You do not, however, have to activate the Vector zone layer. You may place streamtraces on any of Tecplot's field layers, or even on bare axes.

There are two main categories of streamtraces:

- Surface line streamtraces, or streamlines.
- Volume streamtraces.

Surface streamtraces are confined to the surface on which they are placed. They can only be placed in zones displayed as a 2- or 3-D surface. If you try to place streamlines in a zone displayed as a 3-D volume, an error dialog appears, and no streamlines are drawn.

Volume streamtraces can be created only in 3-D volume zones (IJK-ordered or FE-volume zones). Volume streamtraces themselves fall into three categories:

- Volume Lines, or volume streamlines.
- Volume Ribbons, or streamribbons.
- Volume Rods, or streamrods.

You can add streamtraces to your plot either singly or in a rake, which is a set of streamtraces with starting positions along a defined line. Once the rake is added, the individual streamtraces of the rake are identical to singly placed streamtraces.

## 13.1. Creating Surface Streamlines

Surface streamlines include all 2-D streamtraces and 3-D surface streamtraces, which are confined to the surface on which they are placed. You can place streamlines one at a time, or in groups called rakes. A streamline rake is a set of streamlines with starting points along a given line.

You can place streamlines and streamline rakes using either the mouse or the Streamtrace Placement dialog. The mouse enables you to place streamtraces quickly, but the Streamtrace Placement dialog gives you precise control over the starting points.

To create a streamline or streamline rake using the mouse:

1. On the sidebar, choose the Place Streamtrace tool, represented by .
2. If you have not already assigned vector components you will be prompted for them.
3. Move the pointer into the workspace. The pointer changes to a cross-hair.
4. To place a single streamline, click at the desired starting point for the streamline, or Ctrl-click to begin the streamline at the data point nearest to the cross-hair.
5. To create a rake of streamlines, click-and-drag from one end point of the desired rake starting line to the other, then release.

Before placing streamlines, you can change the streamtrace direction or the number of streamtraces per rake using the Streamtrace Placement dialog as discussed below.

To create a streamline or rake of streamlines using the Streamtrace Placement dialog:

1. From the Field menu, choose Streamtrace Placement. If you have not yet assigned vector components for the current frame mode, the Vector Variables dialog appears for you to assign them. Otherwise, the Streamtrace Placement dialog appears, as shown in Figure 13-1.
2. Choose a direction for the streamline integration from the Direction drop-down:
   - **Forward:** The streamline is calculated downstream, that is, in the direction of the flow.
   - **Backward:** The streamline is calculated upstream, that is, against the flow.
   - **Both:** Both the forward and backward streamlines are calculated.

   No matter which direction is chosen for the integration, the arrowheads still point in the forward, “downstream” direction.
3. Under the heading Place Streamtraces by Entering Positions, select either the Use IJK or Use XYZ option. (Use XYZ is the default). If you select Use IJK, also pick a zone from the drop-down labeled Zone.

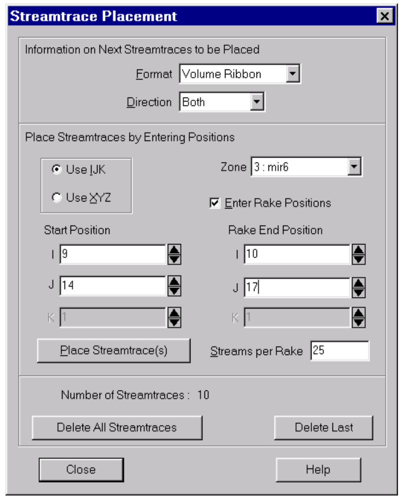



**Figure 13-1.** The Streamtrace Placement dialog.

If you select Use IJK, the fields under Start Position and Rake End Position are labeled I, J, and K. Otherwise, they are labeled X, Y, and Z.

4. Enter the starting position by specifying either a set of IJK-indices or XYZ-coordinates. For finite-element zones, only enter I. The J and K fields will not be available.
5. To place a single streamline, click Place Streamtrace(s).

   To place a rake of streamlines, select the Enter Rake Positions check box and enter the end positions for the rake as either a set of IJK indices or XYZ-coordinates. Then click Place Streamtrace(s). By default, Tecplot draws ten streamlines per rake. To change this, enter a new value in the Streams per Rake field.

   Streamtraces will be terminated at the edge of any cell which is all or partially value-blanked.

Figure 13-2 shows some surface line streamtraces and a streamtrace rake on the cylinder data.

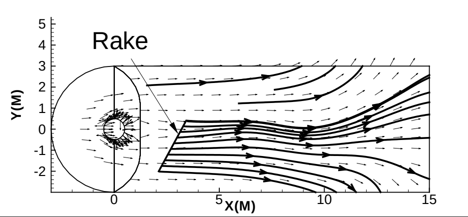



**Figure 13-2.** Surface line streamtraces and a streamtrace rake.

## 13.2. Creating Volume Streamtraces

You can place volume streamtraces using either the mouse or the Streamtrace Placement dialog. The mouse enables you to place streamtraces quickly, but the Streamtrace Placement dialog gives you precise control over the starting points.

Volume streamtraces may only be drawn in 3-D volume zones so you must have at least one volume zone active. It does not matter what plot style you choose for the zone, but you must choose a setting that includes streamtraces in the Volume Objects option on the Plot Attributes dialog's Volume page. The default is to allow streamtraces to be drawn in all zones.

There are a number of different ways to place volume streamtraces. It is best to have some knowledge of the general direction of flow for the velocity field in the zones, so that you may place your streamtraces in a location of interest. Often it is a good idea to start streamtraces from somewhere within the volume, as opposed to starting on the outer surface of the volume. One of the best ways to start a volume streamtrace is to create a slice through the volume, then place the streamtrace starting position on the slice. A slice through a volume zone and a rake of streamtraces starting from the slice is shown in Figure 13-3.

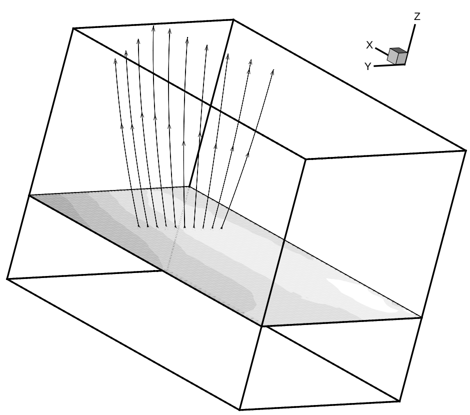
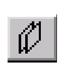



**Figure 13-3.** A slice through a volume zone with a rake of streamlines starting from the slice.

To place volume streamtraces on a slice perform the following steps:

1. Add a slice plane to your volume zone using the Slice tool  from the sidebar. Click on the volume zone to add a slice. You may press I, J, K, X, Y, or Z on your keyboard first to orient the slice in one of the constant I-, J-, K-, X-, Y-, or Z-planes.

2. Click on the Place Streamtrace tool  on the sidebar.
3. Press D on your keyboard to create streamrods, R for streamribbons, or V to create volume streamlines. You may also choose streamtrace types using the Streamtrace Placement dialog from the Field menu.
4. Place a single streamtrace by holding down Alt on your keyboard while clicking on the slice, or place a rake of streamtraces by holding down Alt and clicking-and-dragging. Alt will make sidebar tools operate on any objects except for zones, which in this case is the slice.
5. If you want to place streamtraces starting on the outer boundary of your zone, perform steps 2, 3, and 4 without holding down Alt.

To create a volume streamtrace or a rake of volume streamtraces using the Streamtrace Placement dialog:

1. From the Field menu, choose Streamtrace Placement. If you have not yet assigned vector components for the 3D frame mode, the Vector Variables dialog appears for you to assign vector components. Otherwise, the Streamtrace Placement dialog appears, as shown in Figure 13-1.
2. Choose a format for the streamtrace from the drop-down labeled Format. You can choose Surface Line, Volume Line, Volume Ribbon, or Volume Rod. The default is Volume Line for 3-D volume zones, and Surface Line for all other zones.
3. Choose a direction for the streamtrace integration from the Direction drop-down:
   - **Forward:** The streamtrace is calculated “downstream,” that is, in the direction of the flow.
   - **Backward:** The streamtrace is calculated “upstream,” that is, against the flow.
   - **Both:** Both the forward and backward streamtraces are calculated. You should use this only for surface and volume lines. For ribbons and rods, Tecplot always integrates one step from the starting position before actually plotting or extracting the streamtrace.

   No matter which direction is chosen for the integration, the arrowheads still point in the forward, “downstream” direction.
4. Under the heading Place Streamtraces by Entering Positions, select either the Use IJK or Use XYZ option. (Use XYZ is the default). If you select Use IJK, also pick a zone from the drop-down labeled Zone. If you select Use IJK, the fields under Start Position and Rake End Position are labeled I, J, and K. Otherwise, they are labeled X, Y, and Z.
5. Enter the starting position by specifying either a set of IJK-indices or XYZ-coordinates.
6. To place a single streamtrace, click Place Streamtrace(s). To place a rake of streamtraces, select the Enter Rake Positions check box and enter the end positions for the rake as either a set of IJK-indices or XYZ-coordinates. Then click Place Streamtrace(s). By default, Tecplot draws ten streamtraces per rake. To change this, enter a new value in the Streams per Rake field.

Figure 13-4 shows several examples of volume streamtraces.

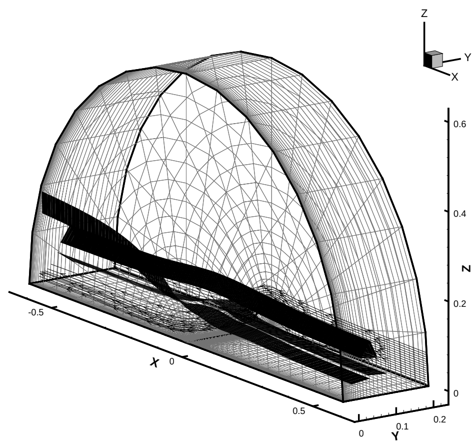



**Figure 13-4.** Volume streamlines, volume ribbons, and volume rods.

## 13.3. Controlling Streamtrace Plot Attributes

You can control the plot attributes, or style, of your streamtraces using the Streamtrace Details dialog. These style attributes affect all streamtraces in the current frame, including those already placed. They do not affect extracted streamtrace zones, discussed in Section 13.7, “Extracting Streamtraces as Zones,” because these are now ordinary ordered zones, not streamtraces at all.

### 13.3.1. Streamlines

The following attributes may be set with the Lines page of the Streamtrace Details dialog, shown in Figure 13-5.

- **Whether streamtraces are displayed:** Select the Show Streamtraces check box if you want streamtraces drawn on your plot. By default, this check box is selected.
- **Line Color:** Enter the color for all streamtraces. You may set the color to Multi-Color to color the streamtraces by the contour variable in the same manner as color flooding. (If the contour variable is not currently defined, the Contour Variable dialog appears so that you can define it.) You can use the Multi-Color option, for example, to color the streamtraces by the local temperature or by the velocity magnitude.

The following attributes affect surface and volume streamlines only:

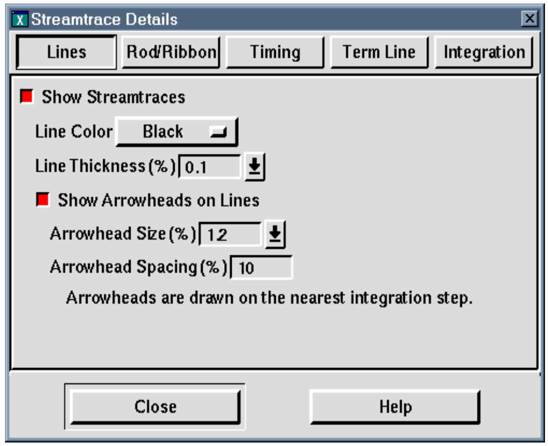



**Figure 13-5.** The Lines page of the Streamtrace Details dialog in Motif.

- **Arrows:** Select the Show Arrowheads on Lines check box to display arrowheads along all streamlines (surface and volume) in the current frame. Arrows are not shown on volume ribbons or volume rods. You can also control the following attributes of the displayed arrows:
  - **Arrowhead Size:** Either enter a value for the arrowhead size (as a percentage of the frame height), or choose a pre-set value from the drop-down menu.
  - **Arrowhead Spacing:** Enter the distance between arrowheads in terms of Y-frame units. A value of ten percent will space arrowheads approximately ten percent of the frame height apart from each other along each streamline.
- **Line Thickness:** Either enter a value for the streamline thickness (as a percentage of the frame height for 2-D lines and as a percentage of the median axis length for 3-D surface lines and volume lines), or choose a pre-set value from the drop-down menu.

## 13.3.2. Streamrods and Streamribbons

The following attributes may be set with the Rod/Ribbon page of the Streamtrace Details dialog, shown in Figure 13-6. They affect volume ribbons and volume rods only:

- **Rod/Ribbon Width:** Enter a width for the volume ribbons and volume rods. The width is expressed in grid units. If you want two sets of streamtraces with different widths, you must create one set and then extract them as zones, then configure a new set of streamtraces with the second width.

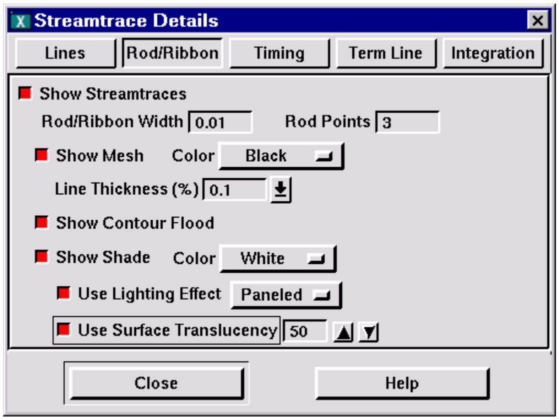



**Figure 13-6.** The Rod/Ribbon page of the Streamtrace Details dialog in Motif.

- **Rod Points:** Volume rods have a polygonal cross-section; this parameter tells Tecplot what that cross-section should be. Three is an equilateral triangle; four is a square; five, a regular pentagon; and so on. If you want two sets of volume rods with different cross-sections, you must create one set and then extract them as zones, then configure a new set of streamtraces with the second cross-section.
- **Show Mesh:** Select this check box to display a mesh.
- **Mesh Color:** Select a mesh color from the drop-down menu, or choose a custom color or multi-color.
- **Mesh Line Thickness:** Select a line thickness from the drop-down menu, or enter your own number in the text field.
- **Show Contour Flood:** Select this check box to display contour flooding.
- **Show Shade:** Select this check box to display shading.
- **Shade Color:** Select a shade color from the drop-down menu, or choose a custom color.
- **Use Lighting Effect:** Select this check box to enable the lighting effect drop-down menu where you may choose Paneled or Gouraud shading.
- **Use Surface Translucency:** Select this check box to enable the surface translucency text field, where you may set the surface translucency from one (opaque) to 99 (translucent).

# 13.4. Deleting Streamtraces

You can delete streamtraces, either individually or in groups, by first selecting them and then either pressing Delete or choosing Clear from the Edit menu. You may delete all streamtraces at once by clicking Delete All Streamtraces on the Streamtrace Placement dialog. You may delete the last streamtrace placed by clicking Delete Last on the Streamtrace Placement dialog.

# 13.5. The Streamtrace Termination Line

A streamtrace termination line is a polyline that terminates any streamtraces that cross it. The termination line is useful for stopping streamtraces before they spiral or stall. If a streamtrace and the termination line intersect on the screen, the streamtrace is considered to cross the termination line. This is true for volume streamtraces within a 3-D volume zone as well as for surface streamlines. Figure 13-7 shows the cylinder data with some streamtraces terminated with a 2-D streamtrace termination line.

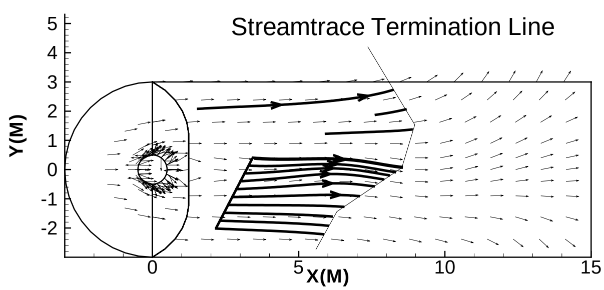



**Figure 13-7.** Surface streamlines and 2-D termination line.

Streamtraces are also terminated whenever any of the following occur:

- The maximum number of integration steps is reached.
- Any point in the streamtrace passes outside the available data.
- The streamtrace enters a cell for which the velocity magnitude is zero.

For more details, see Section 13.8, “Streamtrace Integration.”

## 13.5.1. Creating a Termination Line

To add a streamtrace termination line:

1. From the sidebar, choose the Add Streamtrace Termination Line tool, represented by the  button, or from the Term Line page of the Streamtrace Details dialog, click Draw Stream Term Line.
2. Move the pointer into the workspace. The pointer becomes a cross-hair.
3. Click at the desired starting point for the termination line, then click at additional points to define the desired polyline.
4. To end the termination line, press Esc or select another tool from the sidebar.

Only one termination line can exist at any one time in a given frame. If you draw a second termination line, the first is automatically deleted.

In 2D frame mode, the termination line is drawn in the grid coordinate system and moves with the data as you zoom and translate. In 3D frame mode, unlike most Tecplot objects, the termination line is drawn in the eye coordinate system, which is essentially the plane of your computer screen. This coordinate system also moves with the data as you zoom and translate. If you rotate a 3-D data set after drawing a streamtrace termination line, streamtraces previously terminated by the termination line may be terminated at different places, or not terminated at all if the rotated streamtrace no longer intersects the termination line. Figure 13-8 shows a 3-D volume plot with streamribbons and a streamtrace termination line, and how the termination points vary as the plot is rotated. Notice that the termination line itself remains in place on the screen as the plot is rotated.

### 13.5.2. Controlling the Termination Line

You control the streamtrace termination line from the Term Line page of the Streamtrace Details dialog, shown in Figure 13-9.

From the Term Line page, you can control the following attributes of the termination line:

- **Active:** If the Active Termination Line check box is selected, the termination line is active, and any streamtraces that cross it are terminated. You can deselect the check box and redraw the plot to view the unterminated streamtraces.
- **Shown:** If the Show Termination Line check box is selected, the termination line is displayed. You can deselect the check box and redraw the plot so that only the terminated streamtraces are displayed, not the termination line.
- The color, line pattern, pattern length, and line thickness of the termination line.

You can select a termination line with the Selector or Adjustor tool. This allows you to interactively move the line (with the Selector), modify the line (with the Adjustor), or delete the line (with either tool).

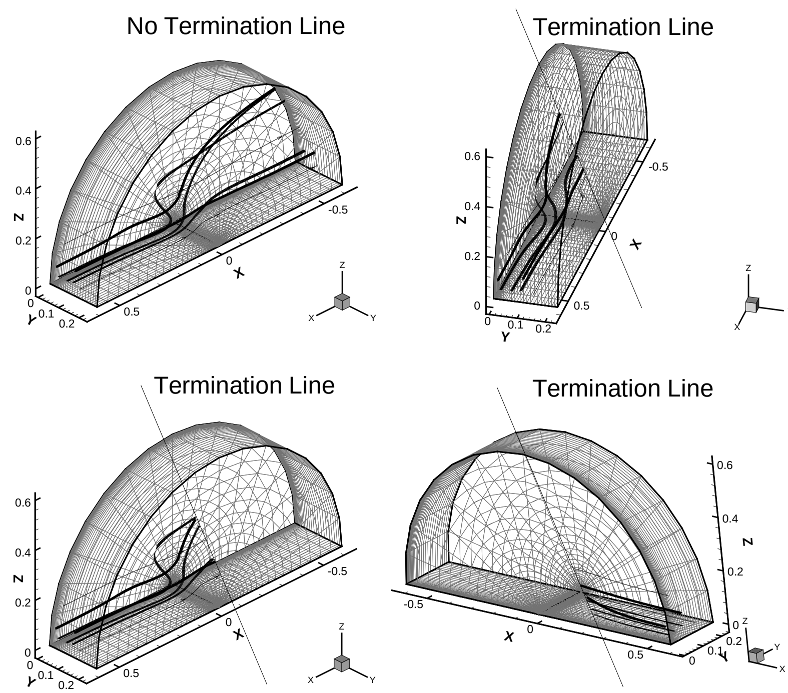



**Figure 13-8.** Terminating volume streamtraces with a termination line.

# 13.6. Streamtrace Timing

Stream markers are symbols plotted along streamtrace paths to identify the positions of particles at certain times. Streamtrace dashes are another means of indicating the passage of time by causing a streamtrace to be “on” for a time interval, then “off” for another time interval. Figure 13-10 shows a plot with both streamtrace markers and dashes.

## 13.6.1. Creating Stream Markers

Stream markers are drawn at timed locations along streamlines. The spacing between stream markers is proportional to the magnitude of the local vector field—that is, it is large in regions where the local magnitude is large, small in regions where the local magnitude is small. You can adjust the spacing between stream markers by specifying the time interval, or delta, between stream markers. Increasing the delta time will increase the space between stream

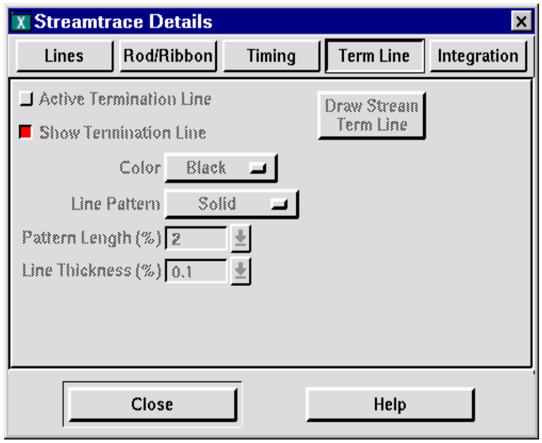



**Figure 13-9.** The Term Line page of the Streamtrace Details dialog in Motif.

markers and vice versa. The actual spacing is the product of the local vector magnitude and the specified delta.

You may also select the shape of your stream marker using the pre-set list under the Shape drop-down menu. Selecting Other from the list activates the Enter ASCII Character option. Clicking this will call up the Enter ASCII Character dialog, where you may enter an ASCII character to be used as your stream marker.

Stream markers are available only for streamlines (surface and volume); they are not available for volume ribbons or volume rods.

To place stream markers along your streamtraces:

1. From the Field menu, choose Streamtrace Details. The Streamtrace Details dialog appears.
2. Click Timing (Timing tab in Windows). The Timing page of the Streamtrace Details dialog appears, as shown in Figure 13-11.
3. Select the check box labeled Show Markers. The three fields immediately below Show Markers become active, as do the fields grouped under the heading Timing.
4. Specify the size, color, and shape of the markers in the fields provided.
5. Specify the timing for the stream markers by entering values in the following fields:
   - **Start Time:** Enter the time at which the first marker is drawn. A start time of zero means that the first marker is drawn at the starting point. A start time of 2.5 means that the first stream marker is drawn 2.5 time units downstream of the starting point.
   - **End Time:** Enter the time after which no more stream markers are drawn.

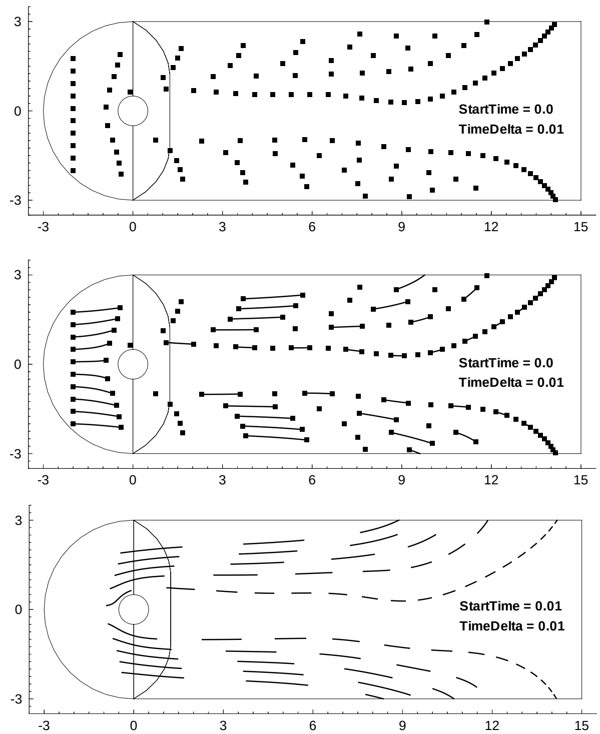



**Figure 13-10.** Streamtrace markers (top), dashes (bottom), and both (middle).

- **Delta Time:** Enter the time interval which measures the time between stream markers. The actual distance between markers is the product of this number and the local vector magnitude.

6. On the sidebar, click Redraw to see the stream markers on existing streamlines; subsequent streamlines will have them drawn automatically.

### 13.6.2. Creating Stream Dashes

Stream dashes, unlike stream markers, are not restricted to streamlines; you can also apply stream dashes to volume ribbons and volume rods. A stream dash shows the streamtrace with a dashed line pattern. The streamtrace is “on” for a time interval, then “off” for a time interval, then “on” again, and so on. The lengths of the dashes and the spaces between them are controlled by the same time delta used for stream markers.

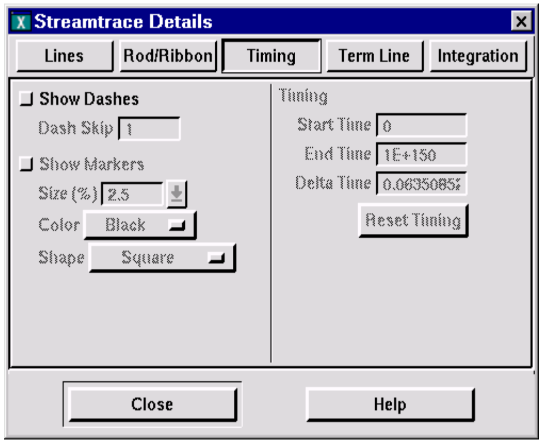



**Figure 13-11.** The Timing page of the Streamtrace Details dialog in Motif.

To use stream dashes:

1. From the Field menu, choose Streamtrace Details. The Streamtrace Details dialog appears.
2. Click Timing (Timing tab in Windows). The Timing page of the Streamtrace Details dialog appears, as shown in Figure 13-11.
3. Select the check box labeled Show Dashes. The Dash Skip field immediately below Show Dashes becomes sensitive, as do the fields grouped under the heading Timing.
4. Enter the dash skip factor, which controls the number of time deltas used for the “off” sections of the streamtraces. Using a dash skip factor of one produces an on-off-on-off pattern. A skip factor of two produces an on-off-off-on-off-off pattern. The actual lengths of the dashes are computed as the product of the delta time and the local vector magnitude.
5. Specify the timing for the stream dashes by entering values in the following fields:
   - **Start Time:** Enter the time at which the first dash is drawn. A start time of zero means that the first dash is drawn at the starting point. A start time of 2.5 means that the first stream dash is drawn 2.5 time units downstream of the starting point.
   - **End Time:** Enter the time after which no more stream dashes are drawn.
   - **Delta Time:** Enter the time interval which controls the length of the dashes. The actual dash length is the product of this number and the local vector magnitude.
6. Click Redraw on the sidebar to see the stream dashes on existing streamtraces; subsequent streamtraces will have them drawn automatically.

## 13.7. Extracting Streamtraces as Zones

In Tecplot you should be able to assign any style you desire to streamtraces without further processing. Temporary streamtrace objects are created and drawn just like zones. However, if you need to make permanent objects from streamtraces you may extract them to zones.

To extract your streamtraces as zones:

1. Create a plot containing streamtraces. You may (if your data includes any 3-D volume zones) use multiple streamtrace formats.
2. From the Data menu, choose Extract, then choose Streamtraces. The Extract Streamtraces dialog appears, containing the single check box Concatenate Common Streamtraces into One Zone.
3. If you want all streamtraces of a given format extracted to a single zone, select the check box labeled Concatenate Common Streamtraces into One Zone. If you select this check box, Tecplot extracts all surface lines into one zone, all volume lines into another, all volume ribbons into a third, and all volume rods into a fourth. Tecplot uses value-blanking to blank out the intervals between streamtraces (and between stream dashes). This is discussed more fully later in this section.
4. If you do not select the check box, each streamtrace is extracted into its own zone.
5. Click Extract to extract the streamtraces to zones. A Working dialog appears while the extraction is proceeding; click Cancel to interrupt the extraction.

Once you have created these new zones, you may treat them as any other zone, and by default, that is what Tecplot does. If you have a mesh plot, you will see the mesh of your original data plotted with the mesh for each of the new zones. You will also see the original streamtraces, which may obscure the plotted streamtrace zones. Once you have extracted the zones, you can delete the original streamtraces by clicking Delete All Streamtraces in the Streamtrace Placement dialog. Figure 13-12 shows some extracted volume ribbon zones, with the original streamtraces deleted.

If timed dashes are active, all extracted streamtraces will be finite-element zones. Otherwise, all extracted streamline zones are I-ordered, and extracted volume ribbon and volume rod zones are IJ-ordered.

## 13.8. Streamtrace Integration

Tecplot uses a predictor-corrector integration algorithm to calculate streamtraces. The basic idea is to create the streamtrace by moving in a series of small steps from the starting point in the direction of, or in opposition to, the local vector field. Each step is only a fraction of a cell or element. Tecplot automatically adjusts the step size based on the local cell shape and vector field variation.

You can control the streamtrace integration by modifying the following parameters in the Integration page of the Streamtrace Details dialog, shown in Figure 13-13:

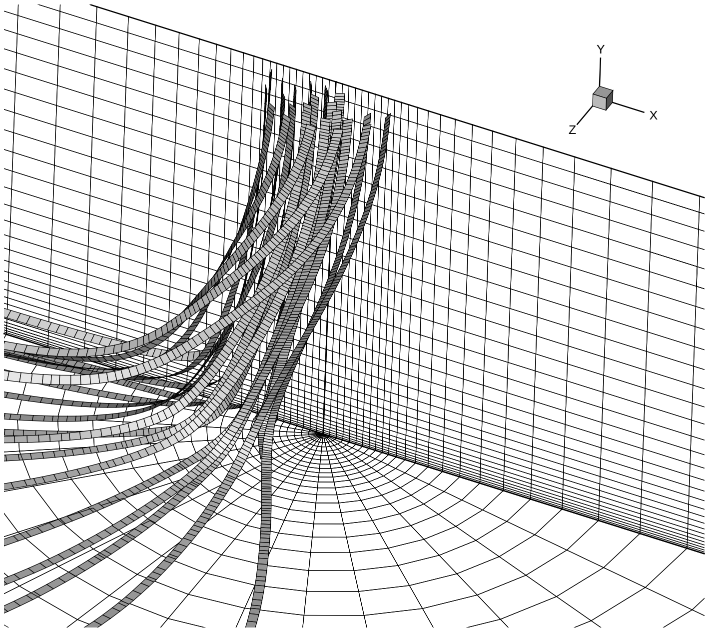



**Figure 13-12.** Extracted volume ribbons.

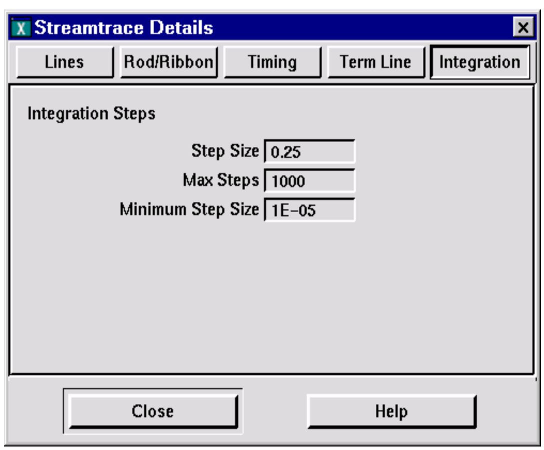



**Figure 13-13.** The Integration page of Streamtrace Details dialog in Motif.

- **Step Size:** Enter the initial and maximum step size Tecplot uses while integrating through the vector field, as a decimal fraction of the local cell or element width. A typical value (and the default) is 0.25, which results in four integration steps through each cell or element. The value for Step Size affects the accuracy of the integration. Setting Step Size too small can result in round-off errors, while setting it too large can result in errors due to missed cells.
- **Max Steps:** Enter the maximum number of steps before the streamtrace is terminated. This prevents streamtraces from spinning forever in a vortex, or from wandering aimlessly in a region where the vector components are very small, very random, or both. If you choose a small Step Size, you should enter a larger Max Steps.
- **Minimum Step Size:** The smallest step size for Tecplot to use. Setting this too small results in integration problems. Setting this greater than or equal to the Step Size results in a constant step size.

During the integration, a streamtrace is terminated if any of the following conditions occur:

- The maximum number of integration steps (Max Steps) have been taken.
- Any point in the streamtrace passes outside the available data. This is a particular concern with volume ribbons: a volume ribbon with a large width may terminate when one edge passes outside the vector field, even though the center is within the field. You can avoid this problem by entering a smaller Ribbon/Rod Width on the Style page of the Streamtrace Details dialog.
- The streamtrace enters a cell for which the velocity magnitude is zero.
- The streamtrace crosses the stream termination line.

# CHAPTER 14 Creating Scatter Plots

Scatter plots are plots of symbols at the data points in a field. The symbols may be sized according to the values of a specified variable, colored by the values of the contour variable, or may be uniformly sized or colored. Unlike contour plots, scatter plots do not require any mesh structure connecting the points, allowing you to make scatter plots of irregular data.

## 14.1. Creating a Scatter Plot

A 2-D scatter plot plots the position of the Y-variable against the position of the X-variable. Thus, it is a representation of the location of data points in the 2-D field. Thought of somewhat differently, the scatter plot is a plot of the vertices of an ordered mesh or the nodes of a finite-element mesh.

To create a scatter plot in Tecplot, you activate the Scatter layer (and deactivate any active layers that you do not want to appear). For example, to create the scatter plot shown in Figure 14-1, do the following:

1. Read in the data file **`simpscat.plt`** from your Tecplot **`examples/dat`** directory. (If you currently have a data set in your Tecplot frame, choose to replace the data set and reset the frame style.)
2. From the sidebar, select the Scatter check box and deselect the Mesh check box (and any other active layer check box).
3. On the sidebar, click Redraw.

## 14.2. Modifying Your Scatter Plot

Once you have read in your data, you can modify your scatter plot attributes using either the Scatter Attributes dialog or the Quick Edit dialog. You can control any of the following attributes from the Scatter Attributes dialog, shown in Figure 14-2:

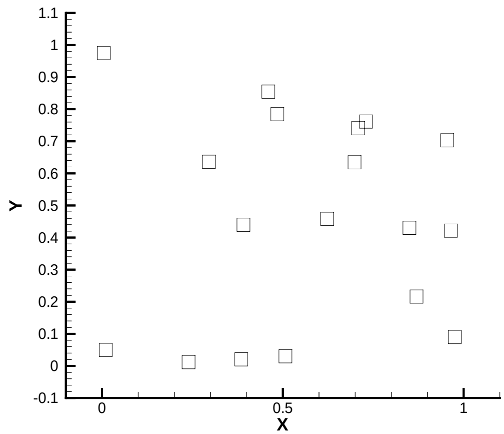



**Figure 14-1.** A scatter plot.

- Which zones are active. See Section 9.3.2, “Controlling Which Zones are Displayed.”
- Whether the scatter symbols are visible for each active zone. See Section 9.3.3, “Controlling Zone Layer Display.”
- The scatter symbol shape. See Section 14.3, “Choosing the Scatter Symbol.”
- The scatter outline color. See Section 14.4.1, “Specifying the Outline Color.”
- The scatter fill color. See Section 14.4.2, “Choosing Filled Symbols and a Fill Color.”
- The scatter symbol size. See Section 14.5, “Specifying Scatter Symbol Size and Font.”
- The scatter line thickness. See Section 9.3.7, “Choosing a Line Thickness.”
- The scatter symbol spacing (Index Skip). See Section 14.6, “Specifying Symbol Spacing.”

The following scatter attributes are assigned on a frame-by-frame basis:

- The scatter-size variable. See Section 14.5, “Specifying Scatter Symbol Size and Font.”
- The optional reference scatter symbol. See Section 14.5.4, “Creating a Reference Scatter Symbol.”
- The scatter legend. See Section 14.7, “Creating a Scatter Legend.”

# 14.3. Choosing the Scatter Symbol

By default, Tecplot uses outlined squares for the scatter symbols. You can choose a different scatter symbol for each zone from any of Tecplot's seven predefined scatter symbols, or any printable character in any of Tecplot's four character sets. You can change the scatter symbol either from the Scatter Attributes dialog or from the Quick Edit dialog.

To choose a scatter symbol:

1. Choose Scatter Attributes from the Scatter sub-menu of the Field menu. The Scatter Attributes dialog appears, as shown in Figure 14-2.

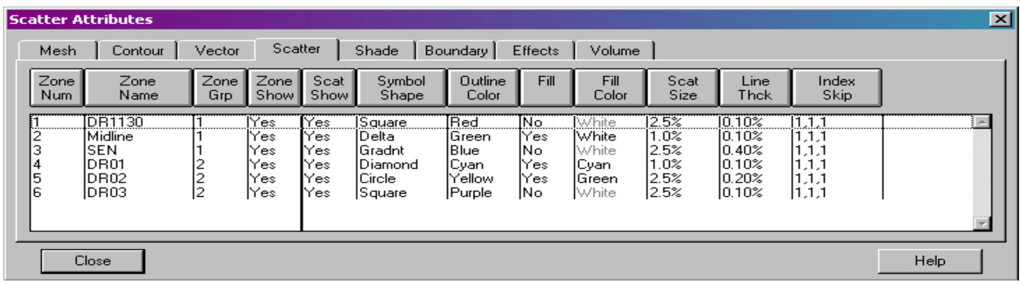



**Figure 14-2.** The Scatter Attributes dialog.

2. Select the zone or zones for which you want to choose a scatter symbol.
3. Click Symbol Shape. A drop-down appears listing the seven predefined scatter symbols together with an Other option. Figure 14-3 shows the seven predefined scatter shapes from the Quick Edit dialog.

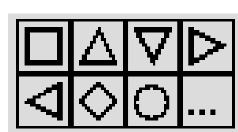


**Figure 14-3.** Pre-defined scatter symbols.

4. Click on the desired symbol shape. If you click on Other, the Enter ASCII Character dialog appears, as shown in Figure 14-4.

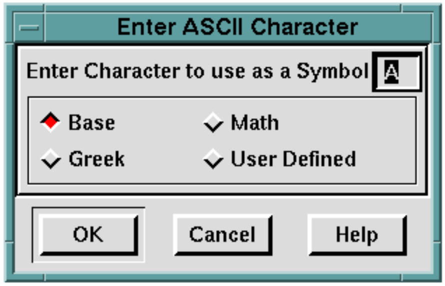
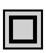
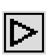
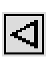
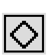
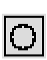
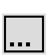



**Figure 14-4.** The Enter ASCII Character dialog.

5. (Other option only) Enter a character to use as a symbol, and then specify the Tecplot character set from which to obtain the symbol: Base (the Courier, Helvetica and Times fonts are collectively referred to as English fonts), Greek, Math, or User Defined. Click OK to dismiss the Enter ASCII Character dialog.

To choose a scatter symbol from the Quick Edit dialog:

1. In the workspace, select the zone or zones for which you want to choose a scatter symbol.
2. On the sidebar, click on Quick Edit to call up the Quick Edit dialog.
3. In the Quick Edit dialog, click on the button for the desired symbol shape, as follows:

-  Plot with square plotting symbols (Square).
-  Plot with upward pointing equilateral triangles (Delta).
-  Plot with downward pointing equilateral triangles (Gradient).
-  Plot with rightward pointing equilateral triangles (Right Triangle).
-  Plot with leftward pointing equilateral triangles (Left Triangle).
-  Plot with diamonds (Diamond).
-  Plot with circles (Circle).
-  Plot with a specified ASCII character from one of Tecplot's four character sets (Other). When you select this option, the Enter ASCII Character dialog appears.

4. (... option only) Enter a character to use as a symbol, and then specify the Tecplot character set from which to obtain the symbol: Base (English Font), Greek, Math, or User Defined. Click OK to close the Enter ASCII Character dialog.

# 14.4. Specifying the Symbol Color

By default, scatter symbols are drawn as outlined symbols, that is, unfilled geometric shapes in a single color. You control the basic color by choosing an outline color for each zone's scatter symbols. You can also specify that a zone's scatter symbols be filled, and then specify a fill color. (If you fill a symbol chosen from the ASCII character set, you obtain a filled rectangle with the character drawn inside in the outline color; the perimeter of the box is also drawn in the outline color.)

For both outline and fill color, one of the available options is Multi-Color, which uses the current contour variable to determine the color of each individual scatter symbol.

## 14.4.1. Specifying the Outline Color

For unfilled scatter symbols, the outline color is simply the symbol color. You can specify a different outline color for each zone using either the Scatter Attributes dialog or the Quick Edit dialog.

To specify an outline color:

1. Choose Scatter Attributes from the Scatter sub-menu of the Field menu. The Scatter Attributes dialog appears.
2. Select the zone or zones for which you want to choose an outline color.
3. Click Outline Color. A drop-down appears listing Tecplot's basic colors together with a multi-color option.
4. Click on the desired color.

or

1. In the workspace, select the zone or zones for which you want to choose a scatter symbol.
2. On the sidebar, click on Quick Edit to bring up the Quick Edit dialog.
3. On the Quick Edit dialog, click Line, then click on the desired color, on M for Multi-Color, or on X to make the line color match the current fill color if the symbols are filled.

If you select the Multi-Color option, each plotting symbol is colored according to the value of the current contour variable at that data point. If no contour variable is currently assigned, the Contour Variable dialog appears with the default contour variable assignment. You can either accept the default or choose a new contour variable.

### 14.4.2. Choosing Filled Symbols and a Fill Color

You can fill any zone's scatter symbols with any of Tecplot's basic colors, or with colors based on the current contour variable. You can control whether a zone's scatter symbols are filled, and if so, what color they are filled with, using either the Scatter Attributes dialog or the Quick Edit dialog.

To specify whether a zone's scatter symbols are filled or unfilled:

1. Choose Scatter Attributes from the Scatter sub-menu of the Field menu. The Scatter Attributes dialog appears.
2. Select the zone or zones for which you want to specify filled or unfilled symbols.
3. Click on Fill. A drop-down appears listing the options Yes and No.
4. Click on Yes for filled symbols, No for unfilled symbols.

or

1. In the workspace, select the zone or zones for which you want to specify filled or unfilled symbols.
2. On the sidebar, click Quick Edit to call up the Quick Edit dialog.
3. Click on the  button for unfilled symbols; click on  for filled symbols.

If you choose filled symbols and want them filled with a color other than the default white, you should also specify a fill color.

To specify a scatter symbol fill color for a zone or zones:

1. Choose Scatter Attributes from the Scatter sub-menu of the Field menu. The Scatter Attributes dialog appears.
2. Select the zone or zones for which you want to specify the fill color.
3. Click Fill Color. A drop-down appears listing Tecplot's basic colors together with the multi-color option.
4. Click on the desired color.

or

1. In the workspace, select the zone or zones for which you want to specify the fill color.
2. On the sidebar, click on Quick Edit to call up the Quick Edit dialog.
3. In the Quick Edit dialog, select the Fill option button, then click on the desired fill color, M for Multi-Color, or X to turn off fill.

If you select the Multi-Color option, each plotting symbol is colored according to the value of the current contour variable at that data point. If no contour variable is currently assigned, the

Contour Variable dialog appears with the default contour variable assignment. You can either accept the default or choose a new contour variable.

## 14.5. Specifying Scatter Symbol Size and Font

You can have Tecplot draw each zone's scatter symbols at a specified size, or you can size each scatter symbol according to the value of a specified variable at that point. To specify fixed-size scatter symbols, you can use either the Scatter Attributes or Quick Edit dialog. To specify variable-size scatter symbols, you must use the Scatter Attributes dialog.

### 14.5.1. Specifying a Fixed Symbol Size

To specify a fixed symbol size:

1. Choose Scatter Attributes from the Scatter sub-menu of the Field menu. The Scatter Attributes dialog appears.
2. Select the zone or zones for which you want to specify the symbol size.
3. Click Scat Size. A drop-down appears listing five preset options, an Enter option, and a Size by Variable option.
4. Click on the desired option. If you click Enter, the Enter Value dialog appears. Continue with Step 5. If you click Size by Variable, the Select Variable dialog appears asking you to choose a scatter-sizing variable. See Section 14.5.2, "Specifying Variable Symbol Sizes," for complete instructions for sizing scatter symbols by variable.
5. (Enter option only) Enter a percentage of the frame height in the Enter Value dialog.

or

1. In the workspace, select the zone or zones for which you want to specify the symbol size.
2. On the sidebar, click Quick Edit to call up the Quick Edit dialog.
3. In the Quick Edit dialog, click Size to the right of the Scatter Symbol buttons. A drop-down appears listing five preset options and an Enter option.
4. Click on the desired option. If you click Enter, the Enter Value dialog appears.
5. (Enter option only) Enter a percentage of the frame height in the Enter Value dialog.

### 14.5.2. Specifying Variable Symbol Sizes

To size the scatter symbols by a variable for a zone or zones:

1. Choose Scatter Size/Font from the Scatter sub-menu of the Field menu. The Scatter Size/Font dialog appears as shown in Figure 14-5.

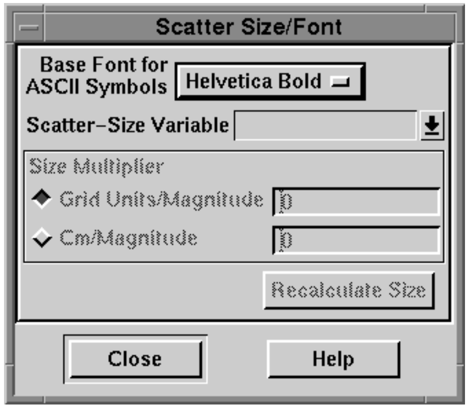



**Figure 14-5.** The Scatter Size/Font dialog.

2. From the drop-down labeled Scatter Size Variable, choose the variable you want to use to size the scatter symbols.
3. (Optional) Specify a size multiplier by selecting one of the two options labeled Grid Units/Magnitude or Cm/Magnitude, then entering a value in the adjacent text field. See Section *14.5.3, "Specifying the Variable Size Multiplier and Font."*
4. Click Close.
5. Choose Scatter Attributes from the Scatter sub-menu of the Field menu. The Scatter Attributes dialog appears.
6. Select the zone or zones for which you want to use variable scatter sizing.
7. Click Scat Size.
8. Click Size by Variable.

Figure 14-6 shows a sample scatter plot sized by a variable.

### 14.5.3. Specifying the Variable Size Multiplier and Font

When you choose Size by Variable for scatter plots, Tecplot multiplies the value of the scatter-size variable at each point by a specified multiplier to determine the actual size of the plotting symbol. You can modify this multiplier from the Scatter Size/Font dialog.

You can change the base font used for ASCII character symbols to any of Tecplot's basic fonts (Helvetica, Courier Bold, etc.). To do this, choose Scatter Size/Font from the Field menu. The Scatter Size/Font dialog appears as shown in Figure 14-5. Choose the desired font from the

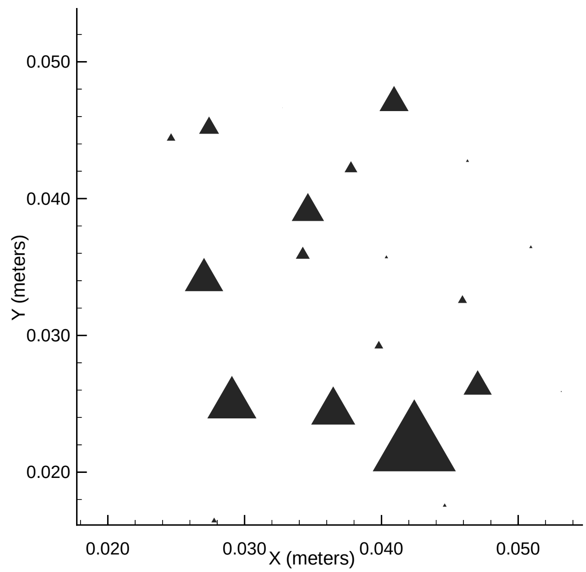



**Figure 14-6.** Scatter plot sized by a variable.

drop-down labeled Base Font for ASCII Symbols. The base font affects all ASCII symbols using the base font in the current frame.

To modify the scatter size multiplier and font:

1. Choose Scatter Attributes from the Scatter sub-menu of the Field menu. The Scatter Attributes dialog appears.
2. Select the zones for which you want to modify the multiplier for scatter symbols.
3. Click Scat Size. On the drop-down which appears, select Size by Variable.
4. Choose Scatter Size/Font from the Scatter sub-menu of the Field menu. The Scatter Size/Font dialog appears, as shown in Figure 14-5.
5. Select Base Font for ASCII Symbols and choose the desired font from the list shown.
6. If you have not specified a scatter-size variable, or if you want to use a different variable, choose a variable from the Scatter Size Variable menu on the Scatter Size/Font dialog. The Size Multiplier region becomes active.
7. Choose the Grid Units/Magnitude or Cm/Magnitude option. By default, Tecplot uses Grid Units/Magnitude. Tecplot calculates and displays an initial size multiplier in the adjacent text field.
8. Enter the desired multiplier in the text field to the right of the selected option.

## 14.5.4. Creating a Reference Scatter Symbol

If you are using a scatter-size variable, it is sometimes useful to create a reference scatter symbol that shows the size at which a data point of a given magnitude will be represented. You create the reference scatter symbol using the Reference Scatter Symbol dialog. The Reference Scatter Symbol appears only if a scatter size variable is defined; if you have not yet created one, select one by choosing Scatter Font/Size from the Field menu, then choosing a Scatter Size Variable from the drop-down.

To create a reference scatter symbol:

1. Choose Reference Scatter Symbol from the Scatter sub-menu of the Field menu. The Reference Scatter Symbol dialog appears, as shown in Figure 14-7.

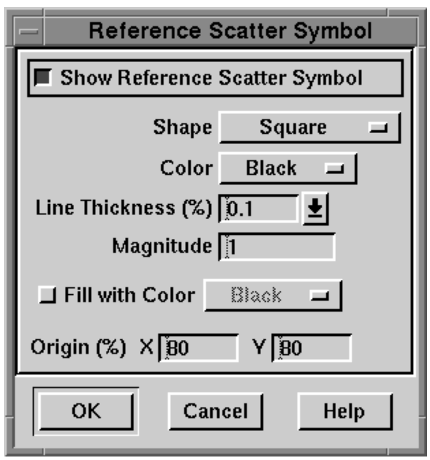



**Figure 14-7.** The Reference Scatter Symbol dialog.

2. Select the check box labeled Show Reference Scatter Symbol.
3. Enter an appropriate value in the Magnitude text box. It may be useful to use the Probe tool  on a symbol of appropriate size, then set the Reference Scatter Symbol magnitude accordingly.
4. Modify the shape, color, line thickness, fill color, and origin as desired.
5. Click OK.

Figure 14-8 shows a scatter plot with a reference scatter symbol.

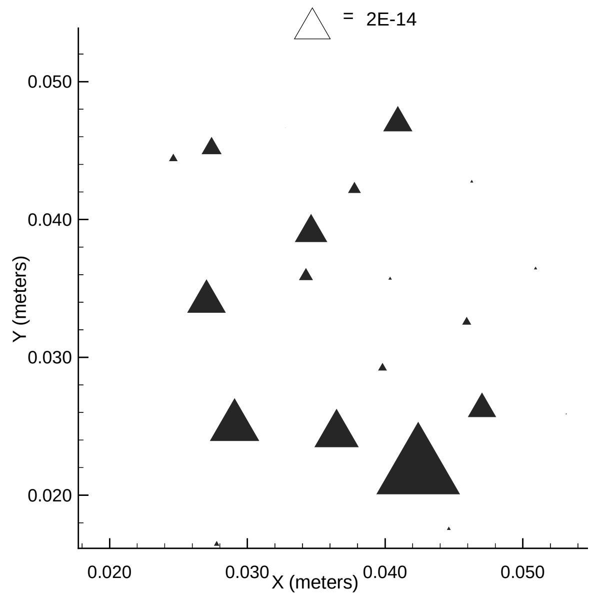



**Figure 14-8.** Scatter plot with reference scatter symbol.

## 14.6. Specifying Symbol Spacing

If your data consists of a dense mesh of points, a scatter plot may be too crowded to be of much use. You can “thin” the scatter plot by plotting only a certain subset of the data points.

You control the number of points plotted with the Index Skip attribute from the Scatter Attributes dialog. For IJ-ordered data, you can specify both an I-skip and a J-skip, while for IJK-ordered data, you can specify I-, J-, and K-skips. (For I-ordered data and finite-element data, only an I-skip is permitted; it allows you to plot every *n*th data point, using the natural order of nodes and data points in the original data set.)

For example, a typical scatter plot from a full-size mesh has so many points it is difficult to see individual symbols (shown in Figure 14-9). Figure 14-10, on the other hand, shows a “thinned” scatter plot of the same data with Index Skip specified, showing every third point in the I-direction and every fourth point in the J-direction.

To specify the symbol spacing:

1. Choose Scatter Attributes from the Scatter sub-menu of the Field menu. The Scatter Attributes dialog appears.

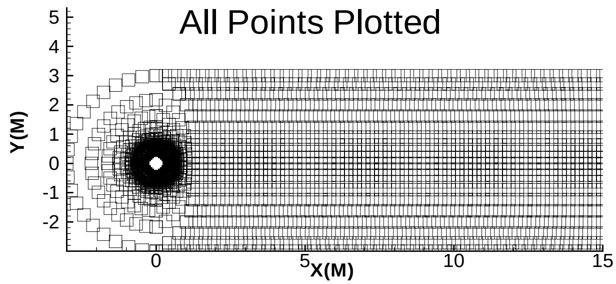



**Figure 14-9.** A crowded scatter plot, using the cylinder data.

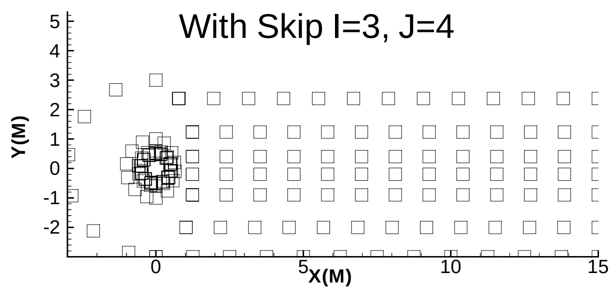



**Figure 14-10.** A scatter plot with Index Skip specified.

2. Select the zone or zones for which you want to specify the symbol spacing.
3. Click Index Skip. A drop-down appears with the options No Skip and Enter Skip. (No Skip is the default.)
4. Click the desired option. If you click Enter Skip, the Enter Index Skipping dialog appears.
5. (Enter Skip only) Enter the desired values of I-Skip, J-Skip, and K-Skip. Figure 14-10 was created with the following settings: *I-Skip=3*, *J-Skip=4*, *K-Skip=1*.

## 14.7. Creating a Scatter Legend

You can generate a legend that shows the style attributes of all scatter symbols. This legend can be positioned anywhere on the plot. You can elect to have the zone names included in the legend.

To create the scatter legend:

1. Choose Scatter Legend from the Scatter sub-menu of the Field menu. The Scatter Legend dialog appears.
2. Select the check box labeled Show Scatter Legend.
3. Select the Show Zone Names check box to include zone names in the legend.
4. Format the text for the legend by choosing a color and font, and specifying the text height as a percentage of the frame height. Enter the desired line spacing in the Line Spacing text field.
5. Specify the location of the upper left corner of the legend by entering values in the X (%) and Y (%) text fields. Enter X as a percentage of the frame width and Y as a percentage of the frame height.
6. Select which kind of box you want drawn around the legend (No Box, Filled, or Plain). If you choose Filled or Plain, format the box using the following controls:
   - **Line Thickness:** Specify the line thickness as a percentage of frame height.
   - **Box Color:** Choose a color for the legend box outline.
   - **Fill Color:** (Filled only) Choose a color for the legend box fill.
   - **Margin:** Specify the margin between the legend text and legend box as a percentage of the text height.

Figure 14-11 shows a scatter plot with a scatter legend.

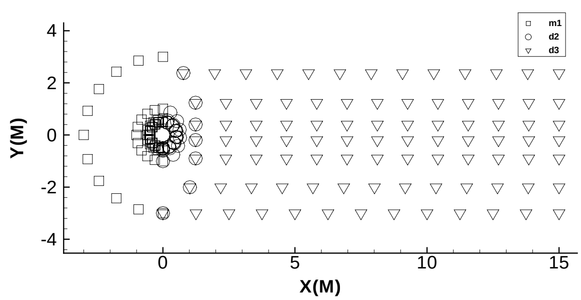



**Figure 14-11.** Scatter plot with legend.

# CHAPTER 15 Creating Shade Plots

Shade plots cover the surface of zones with a single color. In 2-D plots, each shaded zone will have one constant color. In 3-D plots, effects such as translucency and lighting will cause variation in color at different locations on the zone.

Shade plots require IJ- or IJK-ordered, or finite-element data. I-ordered, or irregular data, cannot be used to create shade plots.

## 15.1. Creating 2-D Shade Plots

In 2D frame mode, the only type of shading available is solid zone flooding. Each shaded zone is drawn as a uniform color.

To create a 2-D shade plot:

1. Select the Shade zone layer from the sidebar.

To control the 2-D shade color for a zone or zones:

1. From the Field menu, select Shade Attributes. The Shade Attributes dialog appears as shown in Figure 15-1.
2. In the Shade Attributes dialog, select the zone or zones for which you want to modify the shade color.
3. Click Shade Color. A drop-down appears containing Tecplot’s basic colors.
4. Click on the desired color.

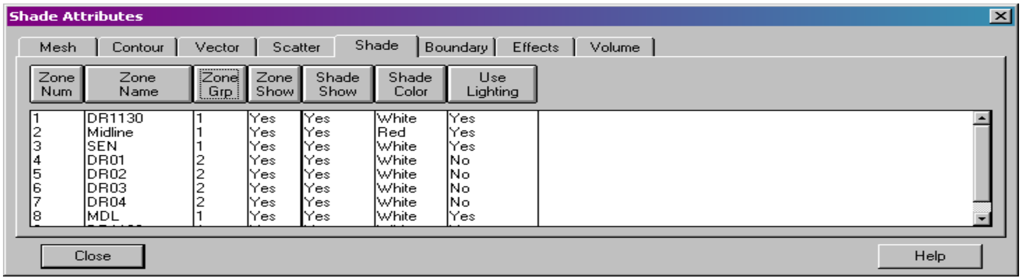



**Figure 15-1.** The Shade Attributes dialog.

## 15.2. Creating 3-D Surface Shade Plots

Three-dimensional shade plots may be created with surface or volume zones. Surface zones, such as IJ-ordered, finite-element triangle or finite-element quadrilateral, are loaded by default into Tecplot as 2-D plots. They may be viewed as 3-D surfaces by selecting 3D frame mode from the sidebar. Volume zones, IJK-ordered, finite-element brick, and finite-element tetrahedral, by default will plot the outer surface with the field layer you select.

To create a 3-D shade plot:

1. Select the Shade zone layer from the sidebar.
2. Control which zones are shaded using the Shade Show option of the Shade Attributes dialog.
3. Control the shade color for each zone using the Shade Color option of the Shade Attributes dialog.
4. Choose which zones will have a 3-D shade effect using the Use Lighting option on the Shade Attributes dialog. To change the lighting effect, go to the Effects tab page of the Shade Attributes dialog.

Figure 15-2 shows a 3-D surface shade plot, with the Use Lighting option selected and the default Paneled lighting effect. The data file from which this plot was generated is delivered with Tecplot as **`demo/plt/spcship.plt`**.

You may specify zone colors for each plotted zone. When Use Lighting is set to No, the zone color is used to uniformly color the zone. For Paneled and Gouraud shading, the zone color is combined with the light source effect, as described in Section 16.1.2, “Lighting.”

To specify the zone color, use the procedures for choosing colors in Section 9.3.4, “Choosing Colors.” (The Multi-Color option is not available for shade plots.)

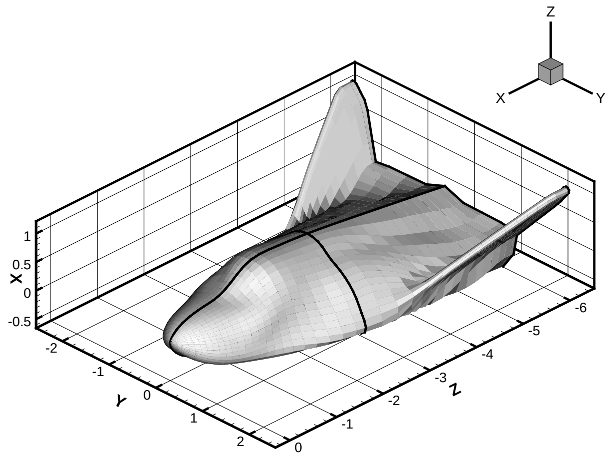



**Figure 15-2.** A 3-D surface shade plot.

# CHAPTER 16 Translucency and Lighting

## 16.1. Translucency and Lighting

The 3-D Effects, Translucency and Lighting, are effective with shaded or contour flooded zones. Each must be selected on the sidebar to be available for zones with a plot. Streamtraces, slices and iso-surfaces all have separate controls from zones for lighting and translucency. The Effect Attributes dialog is shown in Figure 16-1.

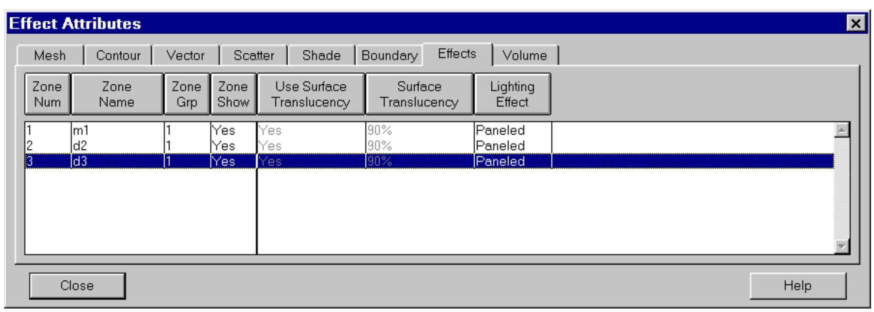



**Figure 16-1.** The Effect Attributes dialog.

### 16.1.1. Translucency

When a zone is translucent, you may view objects inside or beyond the zone. You control the translucency of a zone using the Surface Translucency attribute in the Effects Attributes dialog. Translucency may be set to a value between one, nearly solid, and 99, nearly invisible. There are nine pre-set percentages ranging from ten to 90. You may also use the Enter option to define a percentage of your own. An example of a translucent plot is shown in Figure 16-2.

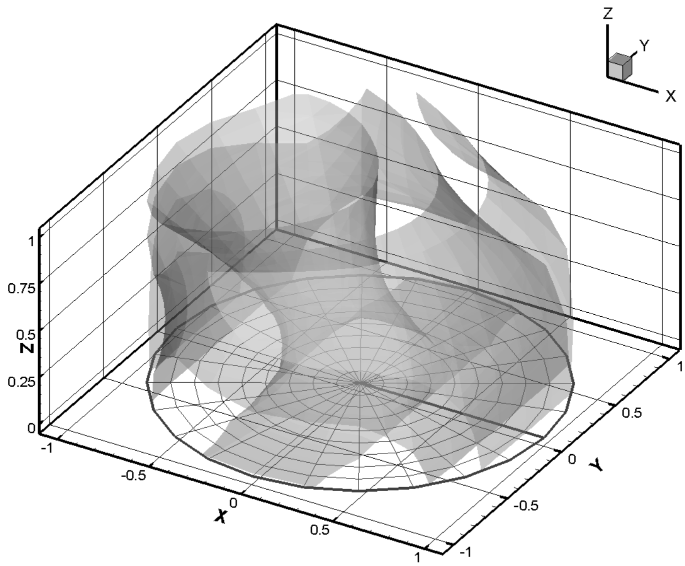



**Figure 16-2.** A plot with translucency.

To create a plot using translucency:

1. Create a 3-D shade or contour flooded plot.
2. Select the Translucency check box on the sidebar.
3. From the Field menu, choose Effect Attributes. The Effect Attributes dialog appears.
4. Select the zone or zones that you want to plot using translucency, then set Use Surface Translucency to Yes.
5. Click Surface Translucency, then select a percentage from the drop-down menu. Alternatively, you may click Enter to type in a value between one and 99.

**16.1.1.1. Plots with Translucency.** All surfaces in 3D frame mode may be made translucent. A different translucency may be assigned to individual zones, and may also be assigned to derived objects such as slices, streamtrace ribbons or rods, and iso-surfaces. Please note that the Translucency check box on the side bar applies only to zones, not slices, streamtraces, or iso-surfaces. Translucency for those objects is controlled through their respective dialogs.

Plots with translucency cannot be printed. Translucency will only appear on your screen, or in exported bitmap images. See Appendix E, “Limits of Tecplot Version 9.0,” for more details.

### 16.1.2. Lighting

There are two types of lighting effects, Paneled and Gouraud.

- **Paneled:** Within each cell the color assigned to each area by shading or contour flooding is tinted by a shade constant across the cell. This shade is based on the orientation of the cell relative to your 3-D light source.
- **Gouraud:** This plot type offers a more continuous and much smoother shading than Paneled shading, but also results in slower plotting and larger print files. Gouraud shading is not continuous across zone boundaries. Gouraud shading is not available for finite-element volume zones when blanking is included. A finite-element volume zone set to use Gouraud shading will revert to Paneled shading when blanking is included.

  IJK-ordered data with Surfaces to Plot set to Exposed Cell Faces, faces exposed by blanking will revert to Paneled shading.

Figure 16-3 shows two shade plots. The one on the left uses a Paneled lighting effect and the one on the right a Gouraud lighting effect.

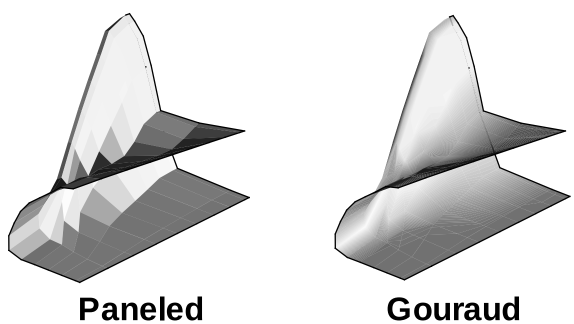



**Figure 16-3.** Plots showing Paneled and Gouraud lighting effects.

**16.1.2.1. 3-D Light Source.** The 3-D light source is a point of light infinitely far from the drawing area. You can specify its location as a point on a hemisphere with a pole along the eye-origin axis. You control the location with the Light Source Position control in the 3D Light Source dialog, shown in Figure 16-4.

The 3-D light source may be different for each frame. By default, the light source is positioned at infinity along the eye-origin axis. The light source position is indicated by a dot over the origin of the 3-D orientation axes displayed in the Light Source Position control.

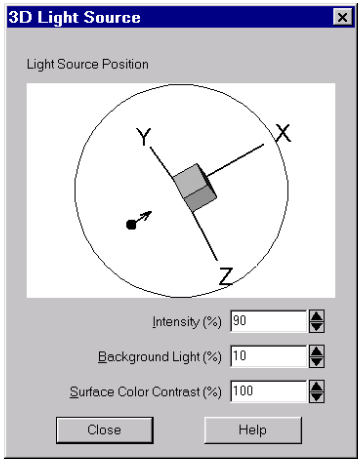



**Figure 16-4.** The 3D Light Source dialog.

To adjust the light source position:

1. From the Field menu, choose 3D Light Source. The 3D Light Source dialog appears.
2. In the Light Source Position control, click the mouse at the desired light source location. You may also drag the mouse to move the light source position. As the light source position moves away from the eye origin ray, its representation appears as an arrow. The length of the arrow indicates how far from the eye origin ray the light source position is. At the eye origin ray, the arrow is pointing directly into the screen, so just a dot is visible; at the horizon (on the circle surrounding the 3-D orientation axes in the Light Source Position control), the arrow is at its longest.

The 3D Light Source dialog also includes the following options:

- **Intensity (%):** Controls the amount of lighting effect produced by the directional light source. An intensity of 100 produces the maximum contrast between lit and unlit areas, and fully lit areas use the full surface color. Lesser values produce less contrast between lit and unlit areas, and fully lit areas use darker colors. An intensity of zero means the light source produces no contrast between lit and unlit areas, and all areas are black.
- **Background Light (%):** Controls the amount of lighting effect applied to all objects regardless of the light source position. A background light of zero means that areas unlit by the directional light source receive no lighting at all and are entirely black, while areas lit by the directional light source get only the effect of that light. Larger values produce more

lighting effect in areas not lit by the directional light source, making these areas show some of the surface color. A background light of 100 means that all areas are lit by the maximum amount, and areas unlit by the directional light source use the full surface color.

**Note:** Intensity and Background Light are cumulative; they can add up to more than 100 and result in colors lightened beyond the base surface color. For example, reds will become pink and grays will become white.

- **Surface Color Contrast (%):** Controls the contrast of the color of light source shaded surfaces before applying lighting effects. A surface color contrast of 100 means that light source shaded surfaces use the full surface color for applying lighting effects. Lesser values mean that the surface color is blended with progressively more white, making light source shaded surface colors lighter. A surface color contrast of zero means that colors are pure white before applying lighting effects; the plot will only be shades of gray.

# CHAPTER 17 Controlling Axes

Tecplot creates axes automatically for all XY-plots and 2- and 3-D field plots. For these automatically created axes, Tecplot determines good tick mark position and spacing, and creates reasonable tick mark and axis labels. You can, however, modify your Tecplot configuration file to change the default behavior, and you can use the Axis menu interactively to exercise virtually complete control over your axes.

You control each axis individually. You specify whether each axis is displayed, its color, position, range, length, tick mark spacing, and many other attributes. You can also specify dependencies between axes that help control the shape of your data when you change the view or individual axis ranges. For 3-D axes, these dependencies can be used to automatically rescale the axes when the X-, Y-, or Z-ranges of your data are significantly different.

Most axis controls are concentrated in the Edit Axis dialog, one page of which is shown in Figure 17-1. As can be gathered from the row of buttons (tabs in the Windows version), this dialog has seven pages. Each page controls a different aspect of the axis. Each page is repeated for each axis.

Tecplot maintains four distinct sets of axes, one for each frame mode. This means that modifying the color, say, of your X-axis line for 2-D plots will not affect 3-D plots. The 2-D X-axis and the 3-D X-axis are different objects.

To edit an axis from the Edit Axis dialog, you must be sure you are working on the correct axis. At the top of each page, except the Area page, there is a check box labeled Show *a*-Axis, where *a* can be X or Y for 2-D axes, X1 through X5 or Y1 through Y5 for X-Y axes, and X, Y, or Z for 3-D axes. To the right of this check box, there is a row of buttons, X and Y for 2-D axes, X, Y, and Z for 3-D axes, X1 Y1 through X5 Y5 for X-Y axes and X and Y for Sketch frame mode. One of these buttons will always appear selected, and will match the check box label. This button indicates the axis the current page modifies. To edit a different axis, simply click on its button.

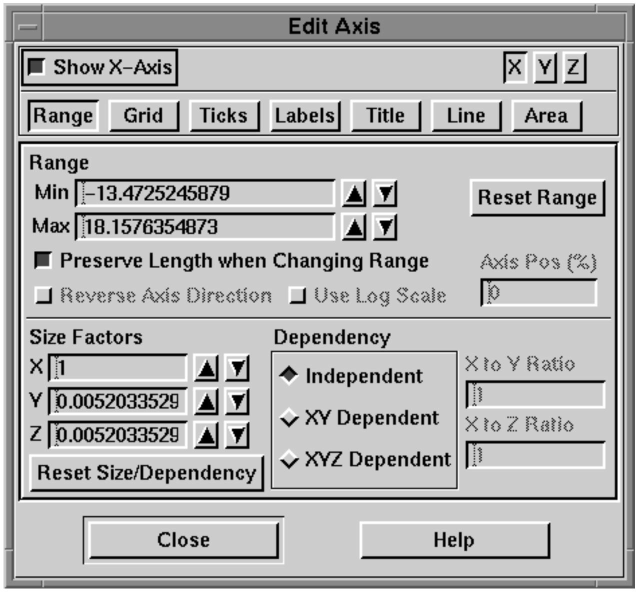



**Figure 17-1.** The Range page of the Edit Axis dialog for a 3-D plot.

## 17.1. Showing and Hiding Axes

The most basic axis control is whether or not to show the axis. Showing an axis, by default, shows the axis line, tick marks, tick mark labels, and axis title for the axis. It is possible to disable any of these components separately, even the axis line. But if you choose not to show an axis, none of the plot components associated with that axis (line, tick marks, tick mark labels, title, or grid lines) is displayed. You can control whether an axis is shown from any page (except Area) of the Edit Axis dialog, using the Show a-Axis check box.

To control the display of an axis, use the following procedure:

1. From the Axis menu, choose Edit. The Edit Axis dialog appears.
2. At the top of the Edit Axis dialog, click X, Y, or Z to select the axis you want to show or hide. The check box to the left of the X, Y, Z buttons is labeled Show *a*-Axis, where *a* is one of X, Y, or Z. For example, by default X is selected, and the check box is labeled Show X-Axis.
3. Select the Show *a*-Axis check box to show the axis; deselect the check box to hide the axis.

You can display axes in Sketch frame mode as well as windows containing data. By default, sketch window axes are not displayed. Follow the above procedure to display sketch axes.

## 17.2. Assigning Variables to Axes

For 2-D axes, Tecplot initially assigns the first and second variables in the data set to the X- and Y-axes, respectively. For 3-D axes, Tecplot initially assigns the first three variables in the data set to the X-, Y-, and Z-axes respectively.

To change variable assignments for 2-D and 3-D axes:

1. From the Axis menu, choose Assign XYZ. A Select Variables dialog appears, with a drop-down for each available axis listing the data set's variables.
2. Choose one variable for each axis.

For XY-plots, assigning variables to axes is part of defining XY-mappings. See Chapter 8, "XY-Plots," for more information.

## 17.3. Modifying the Axis Range

The range of an axis specifies the minimum and maximum data values displayed along it. The length of an axis is its physical length on the screen or paper. The scale of an axis is the ratio of its length to its range. You can modify the range in two ways: in one, changing the range does not affect the length of the axis, and thus modifies the scale. In the second, changing the range preserves the scale, and thus modifies the length.

To change the range while preserving the axis length:

1. From the Axis menu, choose Edit. The Edit Axis dialog appears.
2. Choose the Range page and the appropriate axis.
3. Select the check box labeled Preserve Length when Changing Range.
4. Enter the desired range in the text fields labeled Min and Max. You may also use the up and down arrows to increase or decrease the displayed values.

To change the range while preserving the axis scale:

1. From the Axis menu, choose Edit. The Edit Axis dialog appears.
2. Choose the Range page and the appropriate axis.
3. Deselect the check box labeled Preserve Length when Changing Range.
4. Enter the desired range in the text fields labeled Min and Max. You may also use the up and down arrows to increase or decrease the displayed values.

Figure 17-2 shows the difference between the two methods of changing the range.

You can also change the length of the axis without changing its scale. In XY and 2D frame modes, you can do this using the Grid Position (%) text fields, which are also used to position

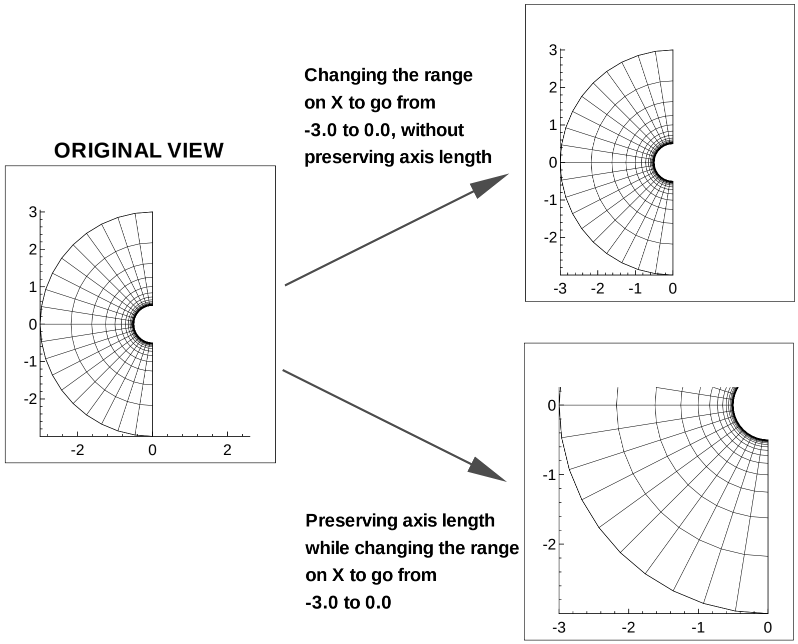



**Figure 17-2.** Preserving length versus preserving scale while changing range.

the grid within the frame. The Left and Right text fields determine the length of the X-axis, and the Top and Bottom text fields determine the length of the Y-axis.

In 3D frame mode, you can change the length of axes using the X, Y, and Z Size Factor fields, but this affects the scale as well. If the 3-D axes are independent (see Section 17.3.1, “Controlling Axis Dependency,” for details), you can resize each axis independently. If the axes are XY-dependent, changing the X or Y size factor changes the other. If the axes are XYZ-dependent, changing one size factor changes the other two.

In XY-plots, you can display the axis range using a logarithmic scale. See Section 8.5.2, “Log Axes.”

### 17.3.1. Controlling Axis Dependency

When you modify the range on one axis, other axes may be affected as well, depending on the current dependency settings for the axes. Axes may be dependent or independent; in general, if axes are dependent, changing the range for one axis will cause similar changes in the other axes, while changing the range of an independent axis will have no effect on the other axes. As you can see, axis dependency within Tecplot is distinct from dependency relations within your data.

For XY-axes and 2-D field plots, the dependency relations Dependent and Independent are the only choices. For XY-plots, independent axes are the default; for 2-D field plots, dependent axes are the default.

For 3-D axes, there are three dependency modes, as follows:

- **Independent:** All axes are independent.
- **XY Dependent:** The X- and Y-axes are dependent upon each other just as for dependent 2-D axes. The Z-axis is independent.
- **XYZ Dependent:** Changing the scale on any axis results in a proportional change in scale on the other two axes, so that the specified X to Y Ratio and X to Z Ratio are preserved.

To control the dependency of your axes:

1. From the Axis menu, choose Edit. The Edit Axis dialog appears.
2. Choose the Range page and the appropriate axis.
3. In the region labeled Dependency, choose one of the available option buttons (for XY- and 2-D plots, Dependent or Independent; for 3-D plots, Independent, XY Dependent, or XYZ Dependent).
4. For 2-D Dependent or either of the 3-D Dependent axis modes, enter the X to Y Ratio in the provided text field. For XY Dependent plots, enter the X1 to Y1 Ratio.
5. For XYZ Dependent mode, also enter the X to Z Ratio in the provided text field.

### 17.3.2. Reversing the Axis Direction

In XY- and 2-D plots, you can reverse the direction on any axis. Normally, the values along an axis increase as you move from bottom to top on the Y-axis, and from left to right on the X-axis. When you reverse the direction, the values increase from top to bottom and right to left.

To reverse the direction of your XY- or 2-D axis:

1. From the Axis menu, choose Edit. The Edit Axis dialog appears.
2. Choose the Range page and the appropriate axis.
3. Select the check box labeled Reverse Axis Direction.

For 3-D axes, you can use negative size factors and/or negative X-to-Y and X-to-Z ratios to reverse the direction of an axis. This can create a left-handed coordinate system, or simply rotate the plot so that, for example, the Z-axis is pointed downward. If the axis mode is Dependent, changing the size factor for one axis changes it on all axes. If the axis mode is XY Dependent, changing the sign of either the X-axis or Y-axis size factor changes the sign of the other. If the axis mode is Independent, changing the sign of the size factor does not affect the signs of the size factors for the other axes, but it does change the sign of the corresponding X-to-Y and X-to-Z ratios.

To reverse the direction of your 3-D axis:

1. From the Axis menu, choose Edit. The Edit Axis dialog appears.
2. Choose the Range page and the appropriate axis.
3. Select the Independent axis mode.
4. Enter a negative number in the Size Factor text field for the axis you want to reverse.

### 17.3.3. Controlling Axis Position

For XY- and 2-D axes, you can control the position of each axis within the frame. You can do this either by specifying a number in the Axis Pos (%) field of the Edit Axis dialog's Range page (which displays the position of the axis as a percentage of the frame width for the Y-axis and as a percentage of the frame height for the X-axis), or by using the Adjustor tool .

To control axis position from the Edit Axis dialog:

1. From the Axis menu, choose Edit. The Edit Axis dialog appears.
2. Choose the Range page and the appropriate axis.
3. Enter the desired position in the Axis Pos (%) text field.

To control axis position with the Adjustor:

1. From the sidebar, choose the  tool.
2. In the workspace, select the axis you want to position. Adjustor handles appear on the axis.
3. Move the pointer over the axis until the cursor changes to a double-sided arrow.
4. Click-and-drag the axis to the desired position. If you drag the axis over the grid area border, the axis will initially stop at the border (this makes it easy for you to position your axis on the edge of the grid area). To continue moving your axis past the border, release the mouse button and continue with Step 3.

## 17.4. Controlling the Axis Grid

The grid area is one or more rectangular regions defined and bounded by the axes. For XY- and 2-D plots, the grid area is simply the rectangle defined by the X- and Y-axes. For 3-D plots, the grid area consists of the three rectangles defined by the X-, Y-, and Z-axes.

The gridlines are a set of lines drawn from one or more axes. Gridlines extend from the tick marks on an axis, across the grid area. Minor gridlines extend from the minor tick marks. Gridlines make it easier to determine the values of individual data points.

The precise dot grid is a set of small square dots drawn at the intersection of every minor gridline. In XY-plots, the axis assignments for the first active XY-map govern the precise dot grid. The precise dot grid option is disabled for 3D frame mode, or when either of the axes for the first active XY-map uses a log scale.

You control the gridlines and precise dot grid from the Grid page of the Edit Axis dialog (Figure 17-3), while grid area attributes are controlled from the Area page.

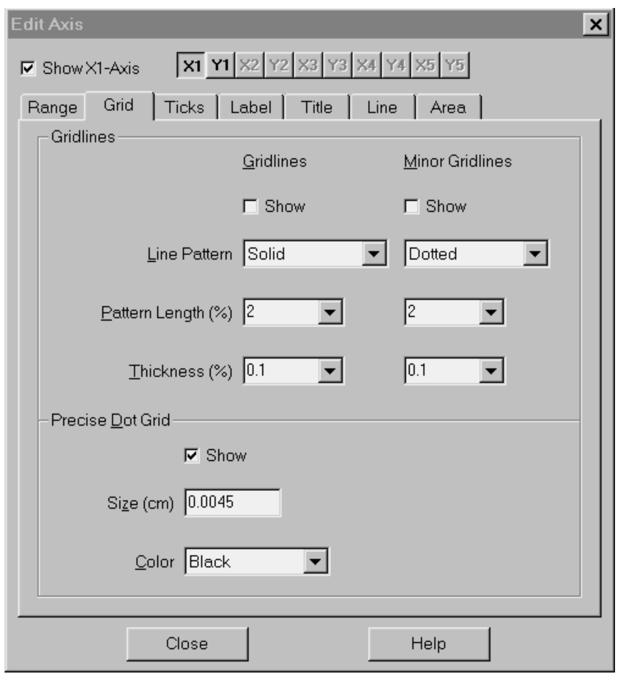



**Figure 17-3.** The Grid page of the Edit Axis dialog.

### 17.4.1. Controlling Gridlines

From the Grid page of the Edit Axis dialog, you can control whether grid lines or minor grid lines are shown, and if so, their line pattern, pattern length, and line thickness. The spacing of grid lines is controlled by the tick mark spacing; see Section 17.5, “Controlling Tick Marks and Tick Mark Labels.”

To control grid lines and minor grid lines:

1. From the Axis menu, choose Edit. The Edit Axis dialog appears.
2. Click Grid (Grid tab in Windows) to call up the Grid page, shown in Figure 17-3.
3. To show gridlines, select the Show check box under Gridlines. To show minor gridlines, select the Show check box under Minor Gridlines.
4. On the Grid page, specify a line pattern, line pattern length, and line thickness for both the gridlines and minor gridlines.

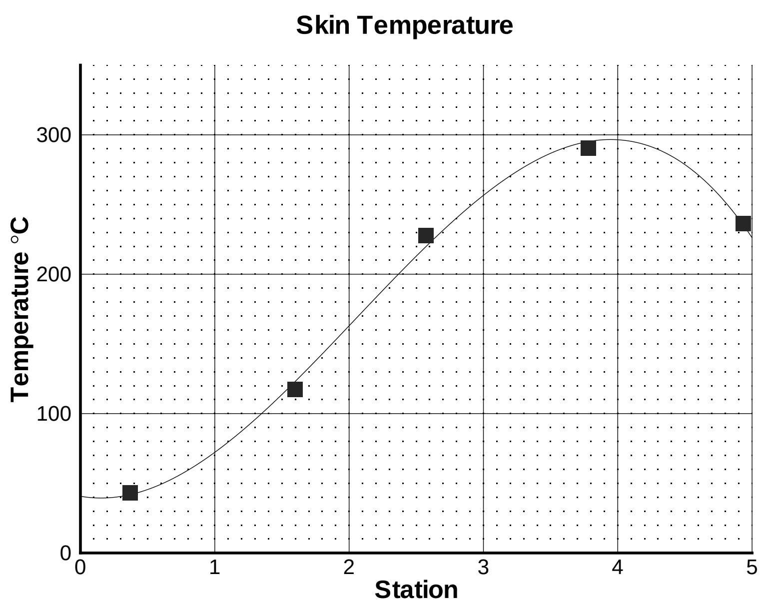



**Figure 17-4.** The effect of turning on the precise dot grid.

### 17.4.2. Controlling the Precise Dot Grid

From the Grid page of the Edit Axis dialog, you can control whether the precise dot grid is shown, and if so, the size and color. Figure 17-4 shows the effect of turning on the precise dot grid.

To control the precise dot grid:

1. From the Axis menu, choose Edit. The Edit Axis dialog appears.
2. Click on the Grid button (Grid tab in Windows) to bring up the Grid page, shown in Figure 17-3.
3. To show the precise dot grid, select the Show check box under Precise Dot Grid.
4. Specify dot size (measured in centimeters on the output) and a color as desired.

### 17.4.3. Controlling the Grid Area

From the Area page of Edit Axis, you control whether the grid area is color-filled, whether the grid area border is displayed, and (except in 3D frame mode) the grid line draw order. For 3-D axes, you can also specify an axis box padding, the minimum distance from the data to the axis box.

To control grid area attributes:

1. From the Axis menu, choose Edit. The Edit Axis dialog appears.
2. Click on the Area button (Area tab in Windows) to call up the Area page, shown in Figure 17-5.
3. To fill the grid area with color, select the check box labeled Fill Behind Grid Area. Select a fill color from the drop-down, or (in 3D frame mode only) select the Use Light Source check box to fill the grid area using the light source color.
4. In 2D frame mode, to display the grid area border, select the Show Border check box, then choose a Border Color and specify a Border Thickness. In 3D frame mode, to display the axis box, select the Show Box check box. The color and line thickness of the 3-D axis box are determined by the axis line color and line thickness, set on the Line page of the Edit Axis dialog. See Section 17.6, “Controlling the Axis Line,” for details.
5. To draw tick marks and tick labels on the grid area border (the default is to draw them only on the axis lines themselves), select the Labels and/or Ticks check box in the Draw on Grid Area Border options. These check boxes control ticks and labels on the sides of the plot away from the main axes. Ticks and labels on the main axes of the plot are controlled on the Ticks and Label pages of the Edit Axis dialog.
6. For XY- and 2-D axes, you may specify a gridline draw order. Gridlines may be drawn either first, before any of the other plotting layers, or last, so that they overlay any plotting layers. Figure 17-6 shows the different effect each option has.

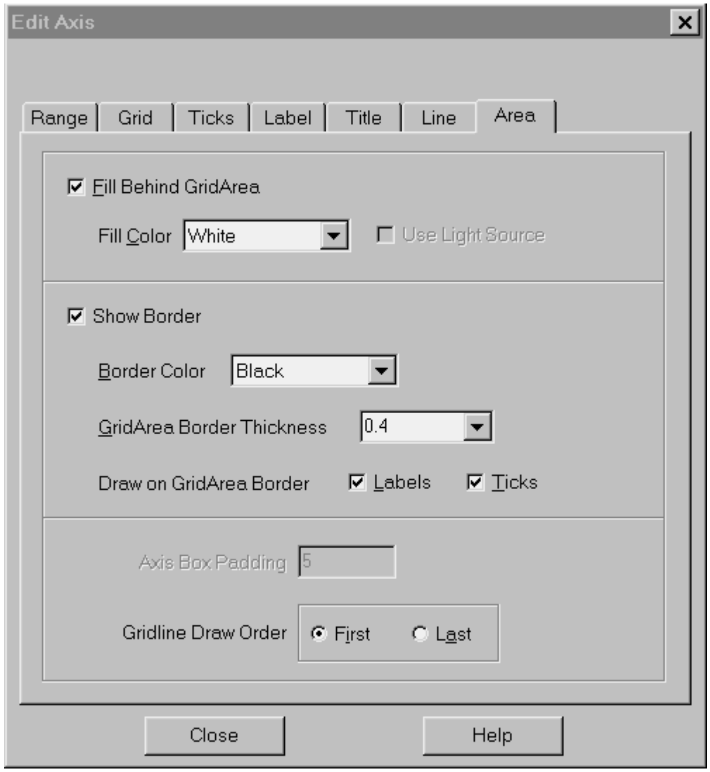



**Figure 17-5.** The Area page of the Edit Axis dialog.

You can also specify the gridline draw order by "pushing" or "popping" the axis grid area from the Edit menu. First select the axis grid area by clicking on a gridline, then choose Push from the Edit menu to plot the gridlines first, or choose Pop to plot the gridlines last.

7. For 3-D axes, you may specify an axis-box padding, which specifies the minimum amount of space between the axis box and the 3-D object as a percentage of the object size. If you decrease the axis-box padding then you must choose 3D Axis Reset from the Axis menu to see the results of your change, otherwise the effect of the new padding will show on the next Redraw.

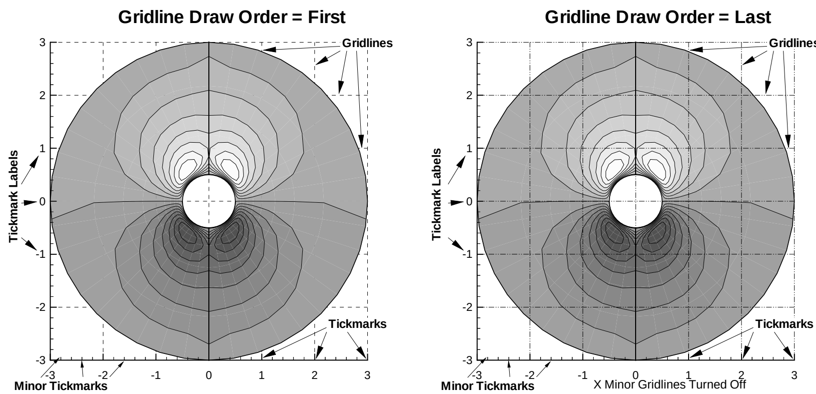



**Figure 17-6.** The grid plotting order.

# 17.5. Controlling Tick Marks and Tick Mark Labels

Each axis can be marked with tick marks, and those tick marks may or may not be labeled, either with numbers or with custom text strings. You control tick marks and their placement using the Ticks page of the Edit Axis dialog. You control the tick mark labels using the Label page of the Edit Axis dialog. You can control the tick mark spacing from either page.

## 17.5.1. Controlling Tick Marks

From the Ticks page of the Edit Axis dialog, you can specify any of the following tick mark attributes independently for each axis:

- Whether tick marks are displayed.
- Length of tick marks and minor tick marks.
- Thickness of tick marks and minor tick marks.
- Number of minor tick marks between tick marks.
- Direction of the tick marks (into the plot, out of the plot, or centered on the axis line).
- Spacing and position of tick marks.

**17.5.1.1. Showing and Hiding Tick Marks.** You can show tick marks on any, all, or none of your axes.

To show tick marks for an axis:

1. From the Axis menu, choose Edit. The Edit Axis dialog appears.
2. Click Ticks (Ticks tab in Windows) to call up the Ticks page, shown in Figure 17-7.

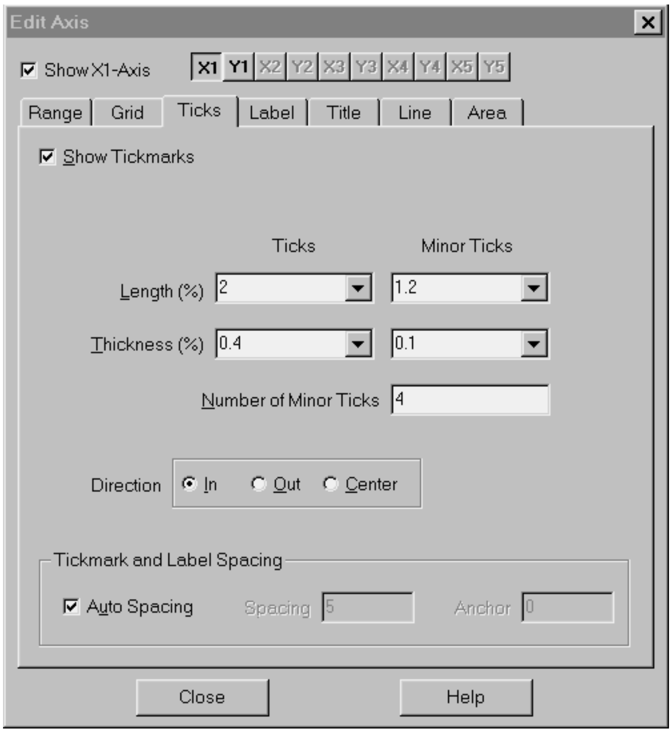



**Figure 17-7.** The Ticks page of the Edit Axis dialog.

3. Choose the axis for which you want to show tick marks. Select the Show Tick Marks check box.

**Note:** There is no separate control for showing minor tick marks, as there is for minor gridlines. To show no minor tick marks, enter zero in the Number of Minor Ticks text field.

To hide tick marks, deselect the Show Tick Marks check box.

**17.5.1.2. Controlling Tick Mark Length.** You can specify length independently for the tick marks and minor tick marks on each axis. You specify length as a percentage of the frame height for XY- and 2-D axes, and as a percentage of the median axis length for 3-D axes.

To specify tick mark and minor tick mark length:

1. From the Axis menu, choose Edit. The Edit Axis dialog appears.
2. Choose the axis for which you want to specify tick mark length.
3. Click Ticks (Ticks tab in Windows) to call up the Ticks page.
4. Enter a length for ticks and minor ticks in the provided text fields, or choose a preset value from the adjacent drop-downs.

**17.5.1.3. Controlling Tick Mark Thickness.** You can specify line thickness independently for the tick marks and minor tick marks on each axis. You specify line thickness as a percentage of the frame height.

To specify tick mark and minor tick mark line thickness:

1. From the Axis menu, choose Edit. The Edit Axis dialog appears.
2. Click Ticks button (Ticks tab in Windows) to call up the Ticks page.
3. Choose the axis for which you want to specify tick mark thickness.
4. Enter a thickness for ticks and minor ticks in the provided text fields, or choose a preset value from the adjacent drop-downs.

**17.5.1.4. Controlling the Number of Minor Tick Marks.** By default, Tecplot draws four minor tick marks between each pair of tick marks, giving five subintervals between tick marks. You can modify the number of minor tick marks, and hence the number of subintervals, by entering a new number in the text field labeled Number of Minor Ticks.

**17.5.1.5. Controlling Tick Mark Direction.** You can specify the direction in which the tick marks are drawn, using one of the following options:

- **In:** Tick marks and minor tick marks are drawn from the axis toward the center of the plotting region.
- **Out:** Tick marks and minor tick marks are drawn from the axis away from the center of the plotting region.
- **Center:** Tick marks and minor tick marks are centered on the axis line.

To specify tick mark direction:

1. From the Axis menu, choose Edit. The Edit Axis dialog appears.
2. Click Ticks (Ticks tab in Windows) to call up the Ticks page.
3. Choose the axis for which you want to specify tick mark length.
4. Choose one of the Direction option buttons: In, Out, or Center.

**17.5.1.6. Controlling Tick Mark and Label Spacing.** You can control tick mark and tick mark label spacing directly, or you can use Auto Spacing (the default). When you use Auto Spacing, Tecplot calculates an optimal spacing and number format for your tick marks and tick mark labels. As you change views, particularly while zooming, Tecplot recalculates the number format and spacing.

To use Auto Spacing for tick marks and tick mark labels:

1. From the Axis menu, choose Edit. The Edit Axis dialog appears.
2. Click Ticks (Ticks tab in Windows) to call up the Ticks page, or click Labels (Labels tab in Windows) to call up the Labels page.
3. Choose the axis for which you want to use Auto Spacing.
4. In the area labeled Tick Mark and Label Spacing, select the Auto Spacing check box.

For a particular view, you may want to exercise manual control by specifying an anchor position, which specifies a particular value at which one tick mark should be drawn, and a spacing factor; additional tick marks are drawn at intervals spacing apart.

To exercise manual control of tick marks and tick mark labels:

1. From the Axis menu, choose Edit. The Edit Axis dialog appears.
2. Click Ticks (Ticks tab in Windows) to call up the Ticks page, or click Labels (Labels tab in Windows) to call up the Labels page.
3. Choose the axis for which you want to specify tick mark and tick mark label spacing.
4. In the area labeled Tick Mark and Label Spacing, deselect the Auto Spacing check box.
5. In the text field labeled Spacing, specify a spacing in the units of the variable assigned to the axis.
6. In the text field labeled Anchor, specify a value at which you want one tick mark drawn. Additional tick marks are spaced according to the Spacing parameter away from this anchor tick mark.

## 17.5.2. Controlling Tick Mark Labels

From the Labels page of the Edit Axis dialog, shown in Figure 17-8, you can specify the following attributes for tick mark labels for each axis:

- Whether tick mark labels are displayed.
- The color, font, and size of the tick mark labels.
- The offset of the tick mark labels—that is, the distance between the tick mark labels and the axis line (in frame units for XY- and 2-D axes, and as a percentage of the median axis length for 3-D axes).

- The orientation of the tick mark labels. For 2-D axes, this is just the angle the labels make with the horizontal. For 3-D axes, the orientation can be the angle the labels make with the horizontal, or they can be placed parallel or perpendicular to the axis.
- The format of the tick mark labels.
- The position and spacing of the tick marks and tick mark labels.

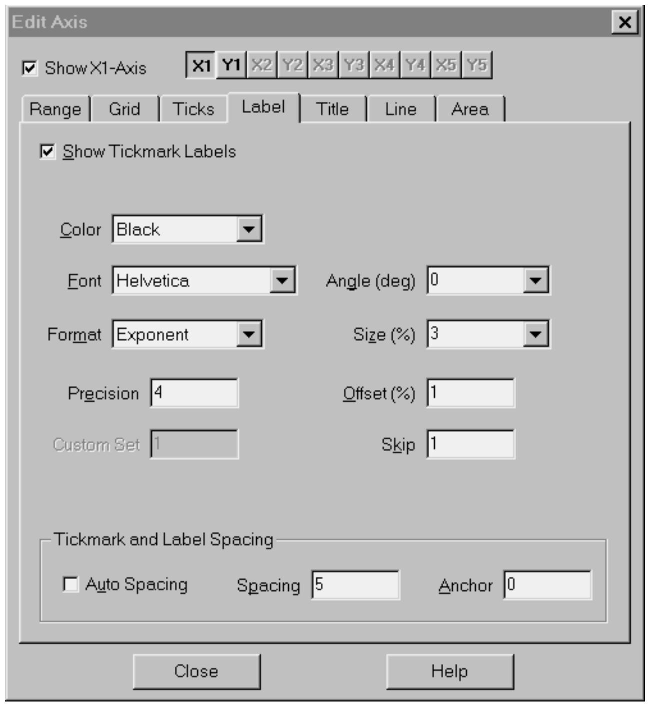



**Figure 17-8.** The Labels page of the Edit Axis dialog.

- A label skip for each axis. This is the number of major tick marks to skip. For example, with the label skip set to 2, every other major tick mark will be labeled.
- You can add labels and/or ticks to all sides of the axis box by using the Show Border and associated options on the Grid page of the Edit Axis dialog.

### 17.5.3. Tick Mark Label Formats

You can choose several numeric formats for your tick mark labels, or specify a set of text strings to use as custom labels. The following numeric formats are available:

- **Integer:** Tick marks are labeled in integer format (for example, 12). If this format is selected, tick mark labels with a decimal part are truncated.
- **Float:** Tick marks are labeled with floating-point numbers (for example, 10.2).
- **Exponent:** Tick marks are labeled using numbers in exponential format (for example, 1.02E-03).
- **Best Float:** Tecplot automatically selects the best floating-point representation of the tick mark labels.
- **Superscript:** Tick marks are labeled with numbers in scientific notation (for example, $1.2 \times 10^{-3}$).

Custom labels are text strings defined in your data file that allow you to print meaningful labels for variables that do not contain numeric data, such as variables that contain names or levels (such as Yes/No, Small/Medium/Large, or months of the year) of a categorical variable. Custom labels are defined using the **`CUSTOMLABELS`** record; each **`CUSTOMLABELS`** record corresponds to one custom set. When you choose custom labels for an axis, you also choose which custom set should be used for that axis.

To specify the format for your tick mark labels:

1. From the Axis menu, choose Edit. The Edit Axis dialog appears.
2. Choose the axis for which you want to specify the format of the tick mark labels.
3. Click Labels (Labels tab in Windows) to call up the Labels page.
4. Choose one of the five numeric formats Integer, Float, Exponent, Best Float, or Superscript, or Custom from the drop-down labeled Format.
5. (Float, Exponent, or Superscript only) Enter a value in the Precision field to specify the number of digits to the right of the decimal point in the tick mark labels. The default is four.
6. (Custom only) Enter a value in the Custom Set field to specify which set of custom labels to use on this axis.

As a simple example of using custom labels, consider the following data file, containing data about attendance at two schools:

```
VARIABLES="SCHOOL", "ATTENDANCE"
CUSTOMLABELS "Cleveland", "Garfield"
ZONE T="1991"
1 950
2 640
ZONE T="1992"
1 1010
2 820
```

The numbers 1 and 2 represent the school number, and the **CUSTOMLABELS** record defines Cleveland as school one and Garfield as school two. Once you assign custom labels in Tecplot, the School axis is labeled with “Cleveland” and “Garfield” rather than “1” and “2.”

To create a plot with custom labels:

1. Create a data file with one or more **CUSTOMLABELS** records, and one or more variables with ordered integer values 1, 2, 3, and so forth. The first string in the **CUSTOMLABELS** record corresponds to a value of 1, the second string to 2, and so on.
2. Read the data file into Tecplot.
3. Create a plot. XY-plots are the most likely to use custom labels, but you can use them anywhere.
4. From the Axis menu, choose Edit, and select the Label page of the Edit Axis dialog.
5. Choose the axis for which you want to assign custom labels, select Custom from the Format drop-down, and choose a set of custom labels for the axis from among all the **CUSTOMLABELS** records in the data file. For this example, edit the X-axis and choose custom set 1.
6. Go to the Ticks page of the Edit Axis dialog and deselect the Auto Spacing check box, then set the spacing to one. (You may also want to set the number of minor ticks to zero.)
7. Go to the Range page of the Edit Axis dialog and set the Min and Max value to 0.5 and 2.5 respectively.
8. Close the Edit Axis dialog, then go to the sidebar. Select the Bars check box and deselect the Lines check box from the plot layers area.

The attendance data are plotted in Figure 17-9.

As another example, consider the following data file containing temperature and rainfall data:

```
VARIABLES="MONTH", "TEMPERATURE", "RAINFALL"
CUSTOMLABELS "Jan", "Feb", "Mar", "Apr", "May", "Jun"
"Jul", "Aug", "Sep", "Oct", "Nov", "Dec"
CUSTOMLABELS "Cold", "Cool", "Warm", "Hot"
CUSTOMLABELS "Dry", "Average", "Wet"
1 1 1
2 1 2
3 2 3
4 2 3
5 3 3
6 3 2
7 4 1
8 4 1
```

This weather data file is plotted in Figure 17-10.

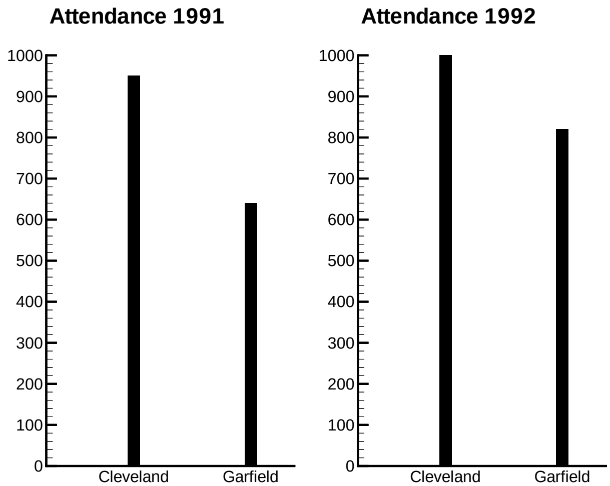



**Figure 17-9.** Bar charts with custom labels.

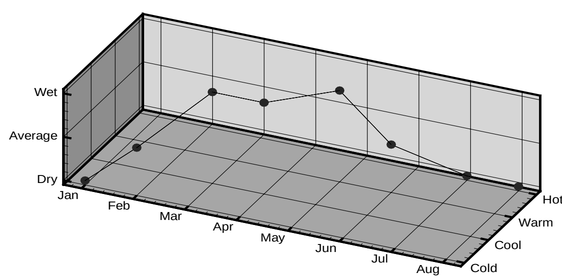



**Figure 17-10.** A 3-D plot with custom labels on each axis.

Custom labels are used cyclically. That is, if the variable assigned to the axis using custom labels goes over the number of custom labels, Tecplot starts with the first label again. This is useful for days of the week, months of the year, or other cyclical data. For example, in the weather data set above, a value of 13 for the **`MONTH`** variable yields a tick mark label of “Jan.” Similarly, a value of five for **`TEMPERATURE`** yields a tick mark label of “Cold”; in this case, that is probably not what you want.

# 17.6. Controlling the Axis Line

The actual axis line is shown by default whenever the axis is shown, but it is just as much an optional part of the plot as the tick marks or tick mark labels. There may be situations when you want to see an axis represented simply by the tick marks or the gridlines, without an additional line for the axis itself. In such cases, you can hide the axis line without turning off the axis as a whole.

To show or hide the axis line:

1. From the Axis menu, choose Edit. The Edit Axis dialog appears.
2. Choose the Line button (Line tab in Windows) to call up the Line page of the Edit Axis dialog, shown in Figure 17-11.

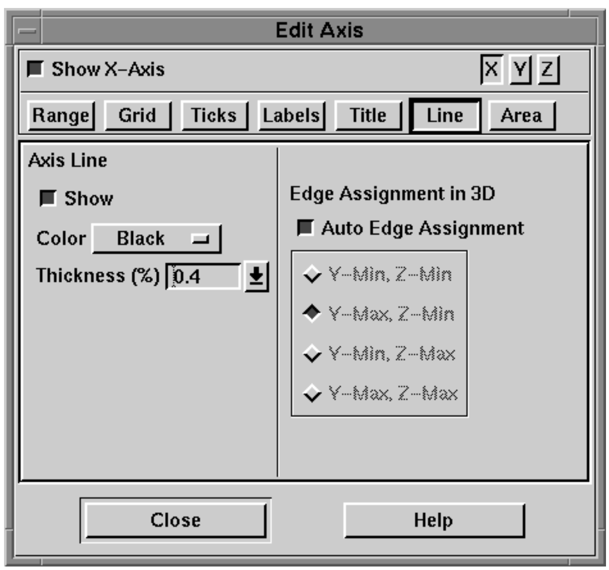



**Figure 17-11.** The Line page of the Edit Axis dialog.

3. Choose the axis for which you want to show or hide the axis line.
4. To show the axis line, select the Show check box. To hide the axis line, deselect the Show check box.

### 17.6.1. Controlling Axis Line Color

You can choose a color for the axis line from the drop-down of Tecplot's basic colors. The axis line color is used to color not only the axis line but also any tick marks or grid lines associated with that axis. In 3-D plots, the color of the axis is also used for the axis box, even if the axis line itself is turned off. The color of tick mark labels, however, is set independently of the axis line color.

To modify the axis line color:

1. From the Axis menu, choose Edit. The Edit Axis dialog appears.
2. Choose the Line button (Line tab in Windows) to call up the Line page of the Edit Axis dialog.
3. Choose the axis for which you want to modify the axis line color
4. Choose one of Tecplot's basic colors from the Color drop-down.

### 17.6.2. Controlling Axis Line Thickness

You can specify the thickness of the axis line. Unlike the choice of color, the axis line thickness affects only the axis line and, in 3-D plots, the axis box; the thickness of gridlines and tick marks is set independently.

To modify the axis line thickness:

1. From the Axis menu, choose Edit. The Edit Axis dialog appears.
2. Choose the Line button (Line tab in Windows) to call up the Line page of the Edit Axis dialog.
3. Choose the axis for which you want to modify the axis line thickness.
4. Enter the desired line thickness (as a percentage of frame height), or choose one of the preset values from the Thickness(%) drop-down.

### 17.6.3. Controlling Edge Assignments in 3-D

In three dimensions, a given axis line might appear in any of four locations relative to the other axes. By default, Tecplot automatically places the three axis lines so they will not interfere with the drawing of the plot, as shown in Figure 17-12.

You can override this automatic edge assignment for each axis line as follows:

1. From the Axis menu, choose Edit. The Edit Axis dialog appears.
2. Choose the Line button (Line tab in Windows) to call up the Line page of the Edit Axis dialog.
3. Choose the axis for which you want to change the edge assignment.

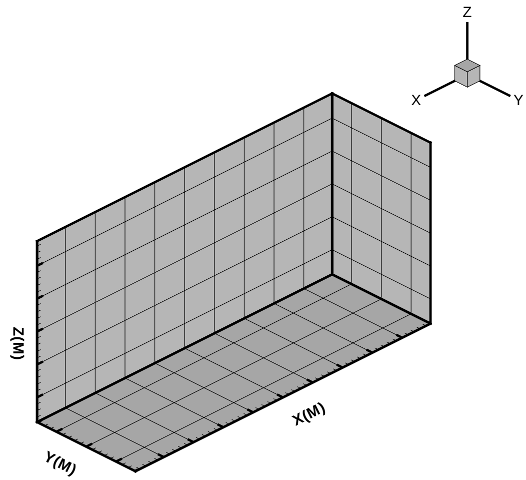



**Figure 17-12.** Default edge positions for 3-D axis lines.

4. Deselect the check box labeled Auto Edge Assignment. Four option buttons, labeled (generically) Min-Min, Max-Min, Min-Max, and Max-Max, become sensitive.
5. Select the option button that yields the desired edge assignment.

# 17.7. Controlling Axis Titles

An axis title is a text label that identifies the axis. By default, Tecplot labels each axis with the name of the variable assigned to that axis. From the Title page of the Edit Axis dialog, you can specify the following attributes for each axis title:

- Whether axis titles are displayed, and if so, what text to use for the title.
- The color, font, and size of the axis title.
- The offset of the axis title—that is, the spacing between the axis title and the axis line.
- The position of the axis title (left, center, or right).

## 17.7.1. Choosing an Axis Title

You have three options in choosing an axis title:

- **No Title:** The axis is untitled.
- **Use Variable Name:** The axis is titled with the name of the variable assigned to the axis.

- **Use Text:** The axis is titled with text that you supply.

To specify the axis title:

1. From the Axis menu, choose Edit. The Edit Axis dialog appears.
2. Choose the Title button (Title tab in Windows) to call up the Title page of the Edit Axis dialog.
3. Choose the axis for which you want to modify the axis title.
4. Select one of the option buttons No Title, Use Variable Name, or Use Text.
5. (Use Text only) If you select Use Text, enter the desired axis title in the text field immediately below the set of option buttons, as shown in Figure 17-13.

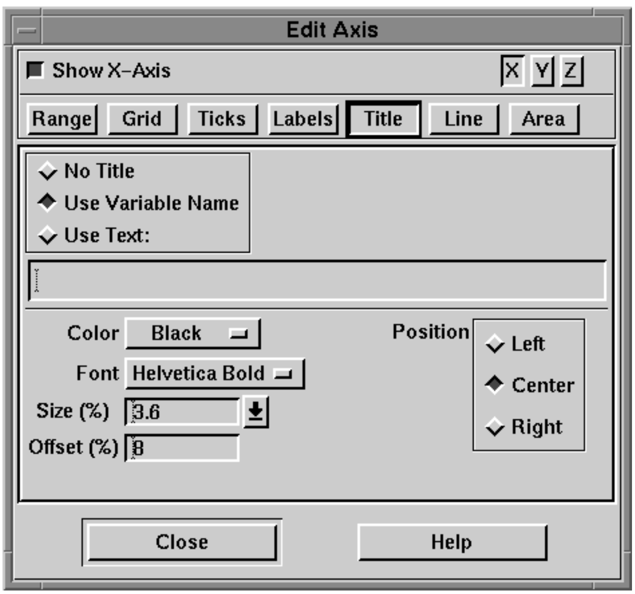



**Figure 17-13.** The Title page of the Edit Axis dialog.

## 17.7.2. Controlling Axis Title Offset

The axis title offset prevents Tecplot from printing your axis title directly on top of the axis. You specify the offset as a percentage of frame height for XY- and 2-D axes, or as a percentage of median axis length for 3-D axes. For XY- and 2-D axes, you can specify a negative offset, which moves the axis title to the other side of the axis (that is, inside the drawing area). For 3-

D axes, negative offsets are treated as positive. Zero offset prints the edge of the axis title on the axis.

To specify an axis title offset:

1. From the Axis menu, choose Edit. The Edit Axis dialog appears.
2. Click the Title button (Title tab in Windows) to bring up the Title page.
3. Choose the axis for which you want to modify the axis title offset.
4. Enter the desired offset in the Offset (%) field.

### 17.7.3. Controlling Axis Title Position

You can choose the position of your axis title from the following three options:

- **Left:** The axis title is positioned at the logical left of the axis (normally at the minimum of the axis range, unless the axis is reversed).
- **Center:** The axis title is centered along the axis.
- **Right:** The axis title is positioned at the logical right of the axis (normally at the maximum of the axis range, unless the axis is reversed)

To specify an axis title position:

1. From the Axis menu, choose Edit. The Edit Axis dialog appears.
2. Click the Title button (Title tab in Windows) to bring up the Title page.
3. Choose the axis for which you want to modify the axis title position.
4. Select a position from one of the option buttons Left, Center, or Right.

CHAPTER 18

# Annotating with Text and Geometries

You can enhance any plot, or create a drawing from scratch, using Tecplot's text and drawing tools. Tecplot provides tools for creating polylines, circles, ellipses, squares, and rectangles, in addition to a text tool for creating titles, labels, and any other text you want. You can annotate plots created in XY, 2D, or 3D frame mode.

Alternatively, pure sketches are created in the "Sketch" frame mode, denoted by the letter "S" on the sidebar. Figure 18-1 shows a sketch created with Tecplot drawing tools.

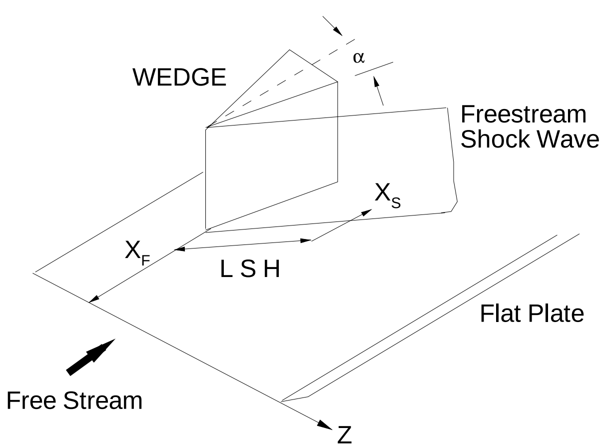



**Figure 18-1.** A sketch created with Tecplot.

## 18.1. Adding Text

Text strings are used for plot titles and labels. Figure 18-2 shows a small sample of the types of text you can create with Tecplot.

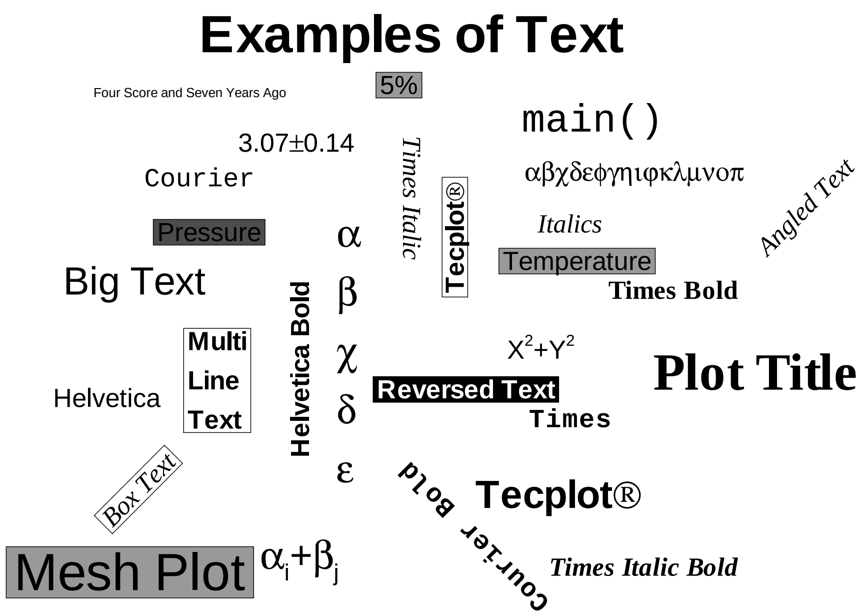
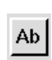



**Figure 18-2.** Text examples created with Tecplot.

To add text to your plot or sketch:

1. From the sidebar, choose the Text tool , or from the Style menu, choose Add Text. In either case, when you move the pointer into the workspace it becomes a cross-hair.
2. Click anywhere in a frame to indicate the location of the text. The Text dialog appears, as shown in Figure 18-3.
3. Enter the desired text in the text area labeled Enter Text String. As you type, the text you enter is echoed in the frame.
4. Modify the text color, font, angle, height, and position as desired. Units for the height and position other than Paper Ruler may be specified by typing them after the number. Use cm for centimeters, in for inches or pix for pixels.

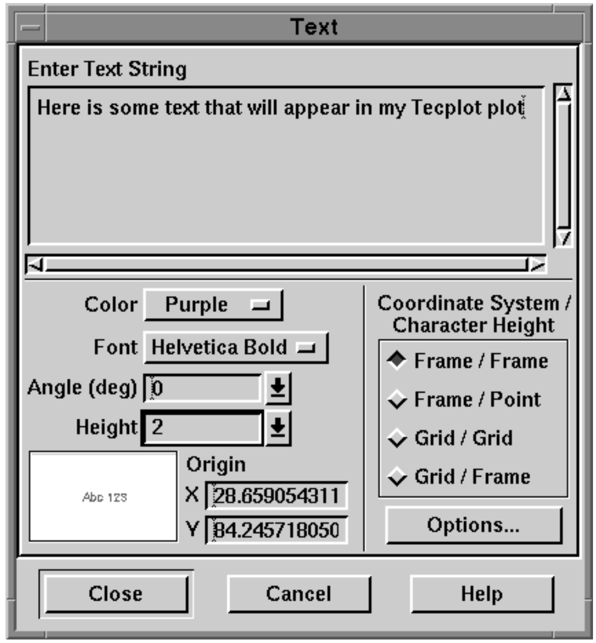



**Figure 18-3.** The Text dialog.

5. Click Options to add a box around your text, modify the line spacing for multi-line text, or set a text anchor location.
6. Click Close to place the text and close the dialog, or click elsewhere in the work area to place additional text.

### 18.1.1. Editing Text

To edit text already placed:

1. From the sidebar, choose the Selector tool .
2. Double-click on the text you wish to edit. The Text dialog appears.
3. Edit the text in the Enter Text String text area, or make any desired modifications to the text attributes (color, font, angle, height, and position). Units for the height and position other than Paper Ruler may be specified by typing them after the number. Use cm for centimeters, in for inches or pix for pixels.
4. Click Options to modify the text box style and the line spacing for multi-line text.

5. To cancel your modifications, click Cancel.

To modify the text attributes of several text items at once, drag the mouse to create a group select rubber band box to enclose the items you want to edit. In the Group Select dialog, deselect all object type check boxes except for Text, and click OK. Then click Quick Edit. The Quick Edit dialog appears. You can use the Quick Edit dialog to modify the font, size, and color of your text items.

### 18.1.2. Deleting Text

To delete text:

1. From the sidebar choose the Selector tool .
2. Select the text you wish to delete, then from the Edit menu, choose Clear, or press Delete. A confirmation dialog appears asking if you are sure you want to clear the selected text.
3. Click OK.

### 18.1.3. Controlling Text Fonts

You can create text using any of Tecplot's eleven built-in fonts. Samples of all eleven fonts are shown in Figure 18-4. Eight of the fonts (Courier, Helvetica and Times) are collectively referred to as English fonts. You can embed Greek, Math, and User-Defined characters into


**Figure 18-4.** Examples of Tecplot's text fonts.

English-font strings by using special lead-in characters called font identifiers, together with the keyboard character that corresponds to the desired character in the chosen font, as shown in Figure 18-5.

The font identifiers and their effects are as follows:

- **‘ (back quote):** Draw the following character from the Greek font.
- **~ (tilde):** Draw the following character from the Math font.
- **@ (at sign):** Draw the following character from the User-Defined font.

For example, to insert the Greek letter Φ into a Tecplot text string, use the combination “`F.” A serif registered trademark symbol (such as that accompanying the word “Tecplot” in Figure 18-2) can be created with the combination “~R.”

Similarly, you can produce subscripts or superscripts by preceding any single character with an underscore (“_”) or caret (“^”), respectively. To subscript or superscript multiple characters, you need to subscript or superscript each character individually. (Tecplot has only one level of superscripts and subscripts; expressions requiring additional levels, such as $e^{x^2}$, must be created by hand using multiple Tecplot text strings.) If you alternate subscripts and superscripts, Tecplot positions the superscript directly above the superscript. Thus, the string **a_b^c** produces $a_b^c$. To produce consecutive superscripts, use the ^ before each superscript character. The string **x^(^a^+^b^)** produces $x^{(a+b)}$ in your plot.

To insert any of the characters “`”, “~”, “@”, “_”, or “^” into text literally, precede the character with a backslash (“\”). To insert a backslash in the text, just type two backslashes (“\\”). In ASCII input files, the number of backslashes must be doubled (two to precede a special character, four to create a backslash) because the Preplot program also requires a backslash to escape special characters.

Embedding and escaping special characters work only in English-font text; they have no effect in text created in Greek, Math, or User-Defined fonts.

### 18.1.4. Using European Characters

Tecplot supports the ISO-Latin one-character encodings. Characters in the ASCII ordinal range from 160-255 are now available, providing support for most of the major European languages.

Figure 18-5 shows the characters supported by Tecplot. Note that the two right-hand columns represent the extended European characters. Using a lead-in character to produce a Greek, Math, or User-Defined character only works with characters in the range 32-126 and is not available for the extended European characters.

| Character Index | English Text | Greek (' prefix) | Math (~ prefix) | User Defined (@) | Character Index | English Text | Greek (' prefix) | Math (~ prefix) | User Defined (@) |
|---|---|---|---|---|---|---|---|---|---|
| 32 | (space) | | | | 80 | P | Π | ∠ | ∅ |
| 33 | ! | ! | ϒ | | 81 | Q | Θ | ∇ | ○ |
| 34 | " | ∀ | ′ | | 82 | R | Ρ | ® | ◇ |
| 35 | # | # | ≤ | | 83 | S | Σ | © | ☼ |
| 36 | $ | ∃ | ⁄ | | 84 | T | Τ | ™ | ☼ |
| 37 | % | % | ∞ | | 85 | U | Υ | ∏ | ☼ |
| 38 | & | & | ƒ | | 86 | V | ς | √ | ☼ |
| 39 | ' | ∋ | ♣ | | 87 | W | Ω | ⋅ | ● |
| 40 | ( | ( | ♦ | | 88 | X | Ξ | ¬ | ● |
| 41 | ) | ) | ♥ | | 89 | Y | Ψ | ∧ | ● |
| 42 | * | ∗ | ♠ | | 90 | Z | Ζ | ∨ | ● |
| 43 | + | + | ↔ | | 91 | [ | [ | ⇔ | |
| 44 | , | , | ← | | 92 | \ | ∴ | ⇐ | |
| 45 | - | − | ↑ | | 93 | ] | ] | ⇑ | |
| 46 | . | . | → | | 94 | ^ | ⊥ | ⇒ | |
| 47 | / | / | ↓ | | 95 | _ | | ⇓ | |
| 48 | 0 | 0 | ° | | 96 | ` | ⎯ | ◊ | |
| 49 | 1 | 1 | ± | | 97 | a | α | 〈 | ● |
| 50 | 2 | 2 | ″ | | 98 | b | β | ® | ● |
| 51 | 3 | 3 | ≥ | | 99 | c | χ | © | ● |
| 52 | 4 | 4 | × | | 100 | d | δ | ™ | ● |
| 53 | 5 | 5 | ∝ | | 101 | e | ε | ∑ | ● |
| 54 | 6 | 6 | ∂ | | 102 | f | φ | ⎛ | ⌽ |
| 55 | 7 | 7 | • | | 103 | g | γ | ⎜ | |
| 56 | 8 | 8 | ÷ | | 104 | h | η | ⎝ | |
| 57 | 9 | 9 | ≠ | | 105 | i | ι | ⎡ | |
| 58 | : | : | ≡ | | 106 | j | φ | ⎢ | |
| 59 | ; | ; | ≈ | | 107 | k | κ | ⎣ | |
| 60 | < | < | … | | 108 | l | λ | ⎧ | |
| 61 | = | = | ⏐ | | 109 | m | μ | ⎨ | |
| 62 | > | > | ⎯ | | 110 | n | ν | ⎩ | |
| 63 | ? | ? | ↵ | | 111 | o | ο | ⎪ | |
| 64 | @ | ≅ | ℵ | | 112 | p | π | | |
| 65 | A | Α | ℑ | | 113 | q | θ | 〉 | |
| 66 | B | Β | ℜ | + | 114 | r | ρ | ∫ | |
| 67 | C | Χ | ℘ | × | 115 | s | σ | ⌠ | |
| 68 | D | Δ | ⊗ | ✳ | 116 | t | τ | ⎮ | |
| 69 | E | Ε | ⊕ | △ | 117 | u | υ | ⌡ | |
| 70 | F | Φ | ∅ | ▽ | 118 | v | ϖ | ⎞ | |
| 71 | G | Γ | ∩ | ⊡ | 119 | w | ω | ⎟ | |
| 72 | H | Η | ∪ | ◈ | 120 | x | ξ | ⎠ | |
| 73 | I | Ι | ⊃ | ⊙ | 121 | y | ψ | ⎤ | |
| 74 | J | ϑ | ⊇ | ⊙ | 122 | z | ζ | ⎥ | |
| 75 | K | Κ | ⊄ | ★ | 123 | { | { | ⎦ | |
| 76 | L | Λ | ⊂ | ∘ | 124 | \| | \| | ⎫ | |
| 77 | M | Μ | ⊆ | + | 125 | } | } | ⎬ | |
| 78 | N | Ν | ∈ | ○ | 126 | ~ | ∼ | ⎭ | |
| 79 | O | Ο | ∉ | ∅ | 127 | | | | |

| Character Index | Extended Character | Character Index | Extended Character |
|---|---|---|---|
| 160 | | 208 | Ð |
| 161 | ¡ | 209 | Ñ |
| 162 | ¢ | 210 | Ò |
| 163 | £ | 211 | Ó |
| 164 | ¤ | 212 | Ô |
| 165 | ¥ | 213 | Õ |
| 166 | ¦ | 214 | Ö |
| 167 | § | 215 | × |
| 168 | ¨ | 216 | Ø |
| 169 | © | 217 | Ù |
| 170 | ª | 218 | Ú |
| 171 | « | 219 | Û |
| 172 | ¬ | 220 | Ü |
| 173 | - | 221 | Ý |
| 174 | ® | 222 | Þ |
| 175 | ¯ | 223 | ß |
| 176 | ° | 224 | à |
| 177 | ± | 225 | á |
| 178 | ² | 226 | â |
| 179 | ³ | 227 | ã |
| 180 | ´ | 228 | ä |
| 181 | µ | 229 | å |
| 182 | ¶ | 230 | æ |
| 183 | · | 231 | ç |
| 184 | ¸ | 232 | è |
| 185 | ¹ | 233 | é |
| 186 | º | 234 | ê |
| 187 | » | 235 | ë |
| 188 | ¼ | 236 | ì |
| 189 | ½ | 237 | í |
| 190 | ¾ | 238 | î |
| 191 | ¿ | 239 | ï |
| 192 | À | 240 | ð |
| 193 | Á | 241 | ñ |
| 194 | Â | 242 | ò |
| 195 | Ã | 243 | ó |
| 196 | Ä | 244 | ô |
| 197 | Å | 245 | õ |
| 198 | Æ | 246 | ö |
| 199 | Ç | 247 | ÷ |
| 200 | È | 248 | ø |
| 201 | É | 249 | ù |
| 202 | Ê | 250 | ú |
| 203 | Ë | 251 | û |
| 204 | Ì | 252 | ü |
| 205 | Í | 253 | ý |
| 206 | Î | 254 | þ |
| 207 | Ï | 255 | ÿ |

**Figure 18-5.** Character indices in Tecplot.

If your keyboard is configured to produce European characters, then the European characters should appear and print automatically with no further setup.

### 18.1.5. Using Character Codes to Generate European Characters

If your keyboard is not configured to produce a specific European character you can generate it by including the sequence *****nnn* in your text where *nnn* the character index value is taken from the character index table found in Figure 18-5. For example, if your keyboard will not generate the é and you want to show the word “latté,” you would enter:

```
latt\233
```

You may need to redraw to get the characters to display clearly.

### 18.1.6. Specifying Text Size and Position

Text can be specified using either of two coordinate systems: frame or grid. In the frame coordinate system, text is positioned relative to the frame, but not to any data that might be in the frame. Thus, if you change the view of the data (for example, by zooming, or translating, or rotating), the frame coordinate text does not move. In the grid coordinate system, on the other hand, the text does move as you alter the view by zooming or translating. If you want to annotate individual data points, it is wise to use grid coordinates.

Once you have chosen a coordinate system, you can choose a set of units for specifying text heights. In the frame coordinate system, you can specify character heights in either frame units (that is, as a percentage of the frame height), or in points. In the grid coordinate system, you can specify character heights in either frame units or grid units (that is, in the same units as shown on the axes; in a sketch plot, the default grid axes run from zero to one in both the X- and Y-direction). Units other than Paper Ruler may be specified by typing them after the number. Use cm for centimeters, in for inches or pix for pixels.

In the Text dialog, you specify the coordinate system and character height units as a pair; you simply choose the desired combination from among the four option buttons under the group label Coordinate System/Character Height: Frame/Frame, Frame/Point, Grid/Grid, or Grid/Frame.

After choosing a Coordinate System/Character Height combination (Frame/Point is the default), you can specify the text height by either choosing a pre-set value or entering a value in the Height field.

**18.1.6.1. Controlling Text Position.** You specify the anchor position for each piece of text by clicking at the desired location in the frame. You have some guidance in this by using the workspace rulers, and you can gain some specific control by using the Snap to Paper or Snap to

Grid sidebar options. However, for complete control over the position of your anchor point, you can specify exact coordinates for the anchor position, or origin of the text, using the Origin controls in the Text dialog.

To specify an exact position for the text anchor position:

1. In the workspace, select the text for which you want to specify an origin.
2. On the sidebar, click Object Details. The Text dialog appears.
3. Enter a value for the X-position in the text field labeled X and a value for the Y-position in the text field labeled Y. Values should be in the coordinate system specified for the text, either frame units or grid units. Units other than Paper Ruler may be specified by typing them after the number. Use cm for centimeters, in for inches, or pix for pixels

The text anchor can be at any of nine locations with respect to the text: vertically, one of Headline, Midline, or Baseline, and horizontally, one of Left, Center, or Right. The anchor location determines whether text is centered about the text origin, or right, left, top, or bottom justified.

To specify the anchor location:

1. In the workspace, select the text for which you want to specify the anchor position.
2. Click Options. The Text Options dialog appears as shown in Figure 18-6.

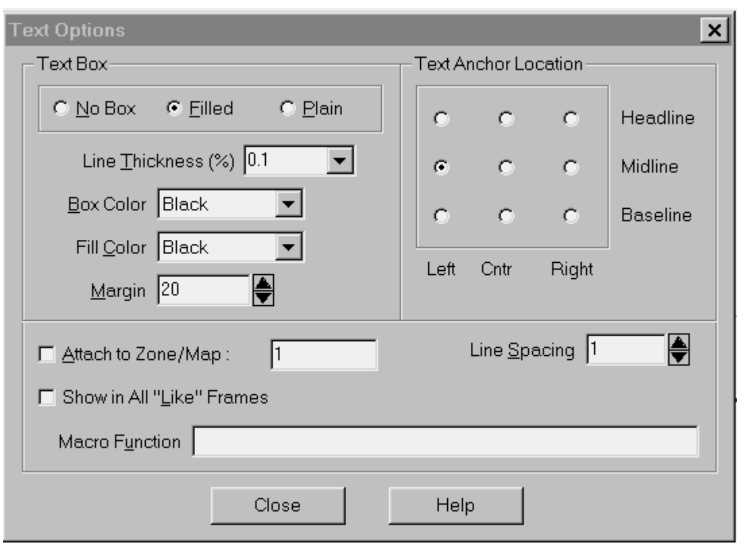



**Figure 18-6.** The Text Options dialog.

3. Under the heading Text Anchor Location, select one of the nine option buttons corresponding to the allowable anchor locations.

**18.1.6.2. Controlling the Text Box.** You can put a plain or filled box around any piece of text. A filled text box obscures all portions of the plot under the text box; a plain box is a transparent outline.

To add a text box:

1. In the workspace, select the text for which you want to specify a text box.
2. Click Options. The Text Options dialog appears.
3. Choose one of the option buttons Filled or Plain.
4. Specify the line thickness and box outline color, and for a filled box, select a fill color.
5. Specify the margin around the text as a percentage of the text character height.

**18.1.6.3. Specifying the Scope of the Text.** By default, text is displayed only in the frame in which it is created. You can, however, choose to have the text appear in all frames using the same data set as the one in which the text was created. Such frames are called like frames.

To propagate text to all like frames:

1. In the workspace, select the text that you want to appear in like frames.
2. On the sidebar, click Object Details. The Text dialog appears.
3. Click Options. The Text Options dialog appears.
4. Select the check box labeled Show in All Like Frames.
5. Click Close.

**18.1.6.4. Attaching Text to Zones or XY-Mappings.** By default, text is always displayed, regardless of which zones or XY-mappings are currently active. Sometimes, however, you use text to highlight a particular feature of a specific zone or XY-mapping, and that text is meaningless unless the zone or XY-map is displayed as well. In such cases, you can control the display of the text by attaching the text to the zone or XY-map.

To attach text to a zone or XY-map:

1. In the workspace, select the text that you want to attach to a zone or XY-map.
2. On the sidebar, click Object Details. The Text dialog appears.
3. Click Options. The Text Options dialog appears.
4. Select the check box labeled Attach to Zone/Map.
5. Enter a zone or XY-mapping number in the text field immediately to the right of the Attach to Zone/Map check box.

6. Click Close.

Once attached to a zone or XY-map, the text is displayed when the zone or XY-map is active, and not displayed when the zone or XY-map is inactive. In XY-plots, this option is controlled by just the XY-mapping number. In 2- and 3-D plots, it is controlled by just the zone number.

### 18.1.7. Adding Dynamic Text

You can add special placeholders to text that changes with the data or the display environment. For example, you can add a date placeholder that Tecplot will replace with the current date at each Redraw. Similarly, you can add a zone name or variable name placeholder.

The complete list of placeholders is as follows:

| Variables | Notes |
|---|---|
| **&(AXISMAXX)** | Maximum value of current X-axis range. |
| **&(AXISMAXY)** | Maximum value of current Y-axis range. |
| **&(AXISMAXZ)** | Maximum value of current Z-axis range. |
| **&(AXISMINX)** | Minimum value of current X-axis range. |
| **&(AXISMINY)** | Minimum value of current Y-axis range. |
| **&(AXISMINZ)** | Minimum value of current Z-axis range. |
| **&(BYTEORDERING)** | Platform’s byte ordering: "**INTEL**" or "**MOTOROLA**" |
| **&(COLORMAPDYNAMIC)** | Returns one if the color map is dynamic, zero if static. |
| **&(DATE)** | Replaced with the current date in the format *dd Mon yyyy*. |
| **&(DATASETFNAME:***nnn***)** | Data set file name of the *nth* file associated with the current data set. If *n* is omitted then all data set file names are show, separated by new lines. |
| **&(DATASETTITLE)** | Replaced with the current data set title. |
| **&(ENDSLICEPOS)** | Replace with the position of the ending slice plane. |
| **&(FRAMEMODE)** | **Zero=Sketch**, **One=XY**, **Two=2D**, **Three=3D**. |
| **&(FRAMENAME)** | Replaced with the current plot. |
| **&(INBATCHMODE)** | A value of one if Tecplot is in batch mode, zero if interactive. |
| **&(ISDATASETAVAILABLE)** | A value of one if a data set exists for the current frame, zero if nonexistent. |

| Variables | Notes |
|---|---|
| **&(ISOSURFACELEVEL:*nnn*)** | Replace with the value of the contour variable on the *nnn*th iso-surface. |
| **&(LAYOUTFNAME)** | Replaced with the name of the current layout file. |
| **&(LOOP)** | Innermost loop counter. |
| **&(MACROFILEPATH)** | Path to the directory containing the most recently opened macro file. |
| **&(MAXB)** | Maximum value for blanking variable. If the frame mode is 2D or 3D, the value is calculated from the current set of active zones. If the frame mode is XY, the value is calculated from the zone assigned to the lowest numbered active XY-mapping. |
| **&(MAXC)** | Maximum value for contour variable. If the frame mode is 2D or 3D, the value is calculated from the current set of active zones. If the frame mode is XY, the value is calculated from the zone assigned to the lowest numbered active XY-mapping. |
| **&(MAXI)** | I-dimension for the lowest numbered active zone for 2D and 3D frame modes. For XY frame mode this represents the maximum I-value for the zone assigned to the lowest numbered active XY-mapping. For finite-element data, this represents the number of nodes in the lowest numbered active zones. |
| **&(MAXJ)** | J-dimension for the lowest numbered active zone for 2D and 3D frame modes. For XY frame mode this represents the maximum J-value for the zone assigned to the lowest numbered active XY-mapping. For finite-element data, this shows the number of elements in the lowest numbered active zone. |
| **&(MAXK)** | K-dimension for the lowest numbered active zone for 2D and 3D frame modes. For XY frame mode this represents the maximum K-value for the zone assigned to the lowest numbered active XY-mapping. For finite-element data, this shows the number of nodes per element for the lowest numbered active zone. |
| **&(MAXS)** | Maximum value for scatter sizing variable for the currently active zones. |
| **&(MAXU)** | Maximum value for variable assigned to the X-vector component for the currently active zones. |

| Variables | Notes |
| --- | --- |
| **&(MAXV)** | Maximum value for variable assigned to the Y-vector component for the currently active zones. |
| **&(MAXVVAR:***nnn***)** | Maximum value of variable *nnn*. |
| **&(MAXW)** | Maximum value for variable assigned to the Z-vector component for the currently active zones. |
| **&(MAXX)** | Maximum value for variable assigned to the X-axis. If the frame mode is 2D or 3D, the value is calculated from the current set of active zones. If the frame mode is XY, the value is calculated from the zone assigned to the lowest numbered active XY-mapping. |
| **&(MAXY)** | Maximum value for variable assigned to the Y-axis. If the frame mode is 2D or 3D, the value is calculated from the current set of active zones. If the frame mode is XY, the value is calculated from the zone assigned to the lowest numbered active XY-mapping. |
| **&(MAXZ)** | Maximum value for variable assigned to the Z-axis for the currently active zones. |
| **&(MINB)** | Minimum value for blanking variable. If the frame mode is 2D or 3D, the value is calculated from the current set of active zones. If the frame mode is XY, the value is calculated from the zone assigned to the lowest numbered active XY-mapping. |
| **&(MINC)** | Minimum value for contour variable. If the frame mode is 2D or 3D, the value is calculated from the current set of active zones. If the frame mode is XY, the value is calculated from the zone assigned to the lowest numbered active XY-mapping. |
| **&(MINS)** | Minimum value for scatter sizing variable for the currently active zones. |
| **&(MINU)** | Minimum value for variable assigned to the X-vector component for the currently active zones. |
| **&(MINV)** | Minimum value for variable assigned to the Y-vector component for the currently active zones. |
| **&(MINVAR:***nnn***)** | Minimum value of variable *nnn*. |
| **&(MINW)** | Minimum value for variable assigned to the Z-vector component for the currently active zones. |

| Variables | Notes |
|---|---|
| **&(MINX)** | Minimum value for variable assigned to the X-axis. If the frame mode is 2D or 3D, the value is calculated from the current set of active zones. If the frame mode is XY, the value is calculated from the zone assigned to the lowest numbered active XY-mapping. |
| **&(MINY)** | Minimum value for variable assigned to the Y-axis. If the frame mode is 2D or 3D, the value is calculated from the current set of active zones. If the frame mode is XY, the value is calculated from the zone assigned to the lowest numbered active XY-mapping. |
| **&(MINZ)** | Minimum value for variable assigned to the Z-axis for the currently active zones. |
| **&(NUMFRAMES)** | Number of frames. |
| **&(NUMPLANES)** | Returns number of graphics bit-planes |
| **&(NUMVARS)** | Number of variables in current data set. |
| **&(NUMWIN)** | Number of frames. (Backward compatibility with Version 6.0.) |
| **&(NUMXYMAPS)** | Number of XY-maps assigned to the current frame. |
| **&(NUMZONES)** | Number of zones in current data set. |
| **&(OPSYS)** | Returns **1=UNIX**, **2=DOS**. |
| **&(PLATFORM)** | Platform name (such as "**SGI**" or "**WINDOWS**"). |
| **&(PLOTTYPE)** | Plot type for the current frame: Zero for Sketch, one for XY Line, two for Cartesian 2D, three for Cartesian 3D, and four for PolarLine. |
| **&(PRINTFNAME)** | Replaced with the name of the current print file. |
| **&(SLICEPLANETYPE)** | Replace with the type of slice plane (X-, Y-, Z-, I-, J- or K-planes). |
| **&(STARTSLICEPOS)** | Replace with the position of the starting slice plane. |
| **&(STREAMSTARTPOS:***nnn***)** | Starting position (X, Y, Z) of the *nnn*th streamtrace. |
| **&(STREAMTYPE:***nnn***)** | Type (Surface Line, Volume Line, Volume Ribbon, Volume Rod) of the *nnn*th streamtrace. |
| **&($***string***)** | Replaced with the value of the system environment variable *string*. |

| Variables | Notes |
|---|---|
| **&(TECHOME)** | Path to the Tecplot home directory. |
| **&(TECPLOTVERSION)** | Returns Tecplot Version. (Currently returns "**90**.") |
| **&(TIME)** | Replaced with the current time in the format *hh:mm:ss*. |
| **&(VARNAME:***nnn***)** | Replaced with the variable name for variable *nnn*. |
| **&(ZONEMESHCOLOR:***nnn***)** | Color of the mesh for the *nnn*th zone. |
| **&(ZONENAME:***nnn***)** | Replaced with the zone name for zone *nnn*. |

The placeholders must be typed exactly as shown, except that the *nnn* in the zone name and variable name placeholders should be replaced by the actual number of the zone or variable, such as **&(ZONENAME:3)** or **&(VARNAME:2)**.

You can, of course, embed the dynamic text strings in text records in a Tecplot-format data file, as in the following example:

```
TEXT CS=FRAME HU=POINT T="&(DATE)"
```

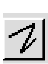
System environment variables can be accessed directly from Tecplot by using the following: **&($***string***)**, where *string* is the name of your environment variable. Using environment variables within Tecplot can add another degree of flexibility by taking advantage of your customized environment. If an environment variable is missing, the environment variable name itself will appear on the screen.

## 18.2. Adding Geometries to Your Plot

Geometries in Tecplot are simply line drawings. Geometries include polylines (a set of line segments), circles, ellipses, rectangles, and squares. Polylines may include arrowheads at either or both ends. Figure 18-7 shows some examples of geometries.

### 18.2.1. Creating Geometries

You create geometries by drawing them in a frame using one of the following sidebar tools:

- The  button to draw polylines. You can also choose this tool by choosing Add Polyline from the Style menu.
- The  button to draw circles. You can also choose this tool by choosing Add Circle from the Style menu.

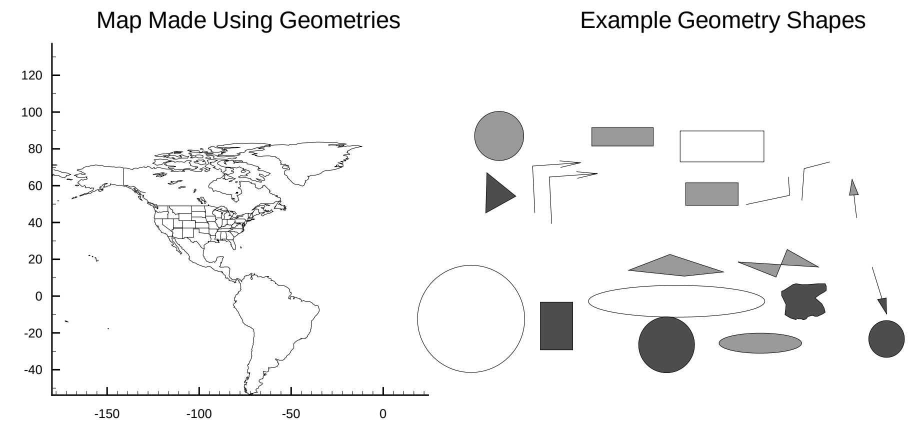
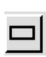



**Figure 18-7.** Sample geometries.

- The  button to draw ellipses. You can also choose this tool by choosing Add Ellipse from the Style menu.
- The  button to draw squares. You can also choose this tool by choosing Add Square from the Style menu.
- The  button to draw rectangles. You can also choose this tool by choosing Add Rectangle from the Style menu.

After choosing any of these tools, when you move the pointer into the workspace, where it becomes a cross-hair. Click once in a frame to set the anchor position for the geometry. To complete the geometry, follow the instructions for each geometry type as follows:

- **For polyline**s: Move the mouse (without dragging) to the desired end point of the first line segment, then click the left mouse button. Move the pointer to the next end point, click, and so on. After placing the last segment, double-click on the final end point or press Esc on your keyboard. To draw a horizontal or vertical line segment, press the H or V keys, respectively, while drawing the segment. After you place the segment's end point, the horizontal or vertical restriction is lifted. To lift the horizon-

tal or vertical line segment restriction without placing the end point, press A on your keyboard. You can draw unconnected line segments in a single polyline; press U on your keyboard to "lift the pen." You can then move the pointer to the start of the next line segment.

- **For circles**: Click at the desired center point of the circle; drag the mouse until the circle is the desired radius, then release.
- **For ellipses**: Click at the desired center point of the ellipse; drag the mouse until the ellipse is the desired size and shape, then release.
- **For squares**: The anchor point of the square is either the lower left-hand corner or the upper right corner of the square. Drag the mouse to the right of the anchor to create a square with the anchor at lower left; drag the mouse to the left to create a square with the anchor at upper right. Release when the square is the desired size.
- **For rectangles**: Drag the mouse until the rectangle is the desired size and shape. In contrast to squares, rectangles can propagate in any direction.

### 18.2.2. Modifying Geometries

You can modify the outline color, line pattern, line thickness, fill color, and position of your geometries, along with other attributes that affect the relationship of the geometry to the frame and the data set (if any). Individual geometry types have their own specific attributes.

**18.2.2.1. Controlling Colors.** Geometries may be filled or unfilled. Unfilled geometries have a single color attribute: the line color. Filled geometries have both a line color and a fill color. You can control the colors from either the Geometry dialog or the Quick Edit dialog.

To specify a geometry's colors:

1. In the workspace, select the geometry or geometries for which you want to modify the color.
2. On the sidebar, click Object Details. If you have selected a single geometry, the Geometry dialog appears. Continue with Step 3. If you have selected multiple geometries, the Quick Edit dialog appears. Continue with Step 4.
3. (Geometry dialog) Specify the geometry's outline color by choosing one of Tecplot's basic colors from the Line Color menu. If you want the geometry filled, select the check box labeled Fill Color, then select a fill color from the menu immediately to the right of the Fill Color check box.
4. (Quick Edit dialog) Specify the outline color for all selected geometries by selecting the Line option button, then clicking on one of the basic colors. If you want the geometries filled, select the Fill option button, then click on one of the basic colors. For both Line and Fill, the M button has no effect on geometries. The X button causes fill to be turned off

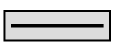
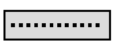
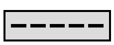
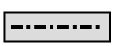
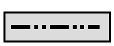
(when Fill option button is selected), or causes line color to match fill color (when Line option button is selected, and fill is present). If no fill is present, the X button also has no effect.

5. Click Close.

**18.2.2.2. Controlling Line Patterns.** The outline of a geometry, or any polyline, can be drawn in any of Tecplot's line patterns. You control the line pattern from either the Geometry dialog or the Quick Edit dialog.

To specify a geometry's line pattern:

1. In the workspace, select the geometry or geometries for which you want to modify the line pattern.
2. On the sidebar, click Object Details. If you have selected a single geometry, the Geometry dialog appears. Continue with Step 3. If you have selected multiple geometries, the Quick Edit dialog appears. Continue with Step 4.
3. (Geometry dialog) Specify the geometry's line pattern by choosing one of Tecplot's six line patterns (Solid, Dashed, Dotted, Dash Dot, Long Dash, and Dash Dot Dot) from the Line Pattern menu. Specify a pattern length by either choosing a pre-set value from the drop-down menu or entering a percentage of the frame height in the text field.
4. (Quick Edit dialog) Specify the line pattern for all selected geometries by selecting the appropriate line pattern button, as follows:

   -  Chooses a solid line.
   -  Chooses a dotted line.
   -  Chooses a dashed line.
   -  Chooses a long dashed line.
   -  Chooses an alternating dot-and-dash line.
   -  Chooses an alternating dash-and-two-dots line.

   Specify a pattern length by clicking on the Pttrn Length button, and choosing either one of the pre-set values or Enter. If you choose Enter, an Enter Value dialog appears; enter the desired pattern length as a percentage of frame height and click OK.
5. Click Close.

**18.2.2.3. Controlling Line Thickness.** You can control the thickness of polylines and the outlines of other geometries. You control the line thickness from either the Geometry dialog or the Quick Edit dialog.

To specify a geometry's line thickness:

1. In the workspace, select the geometry or geometries for which you want to modify the line thickness.
2. On the sidebar, click Object Details. If you have selected a single geometry, the Geometry dialog appears. Continue with Step 3. If you have selected multiple geometries, the Quick Edit dialog appears. Continue with Step 4.
3. (Geometry dialog) Specify the geometry's line thickness by either choosing a pre-set value from the drop-down menu or entering a percentage of the frame height in the Line Thickness (%) field.
4. (Quick Edit dialog) Specify a line thickness by clicking Line Thcknss, and choosing either one of the pre-set values or Enter. If you choose Enter, an Enter Value dialog appears; enter the desired line thickness as a percentage of frame height and click OK.
5. Click Close.

**18.2.2.4. Controlling the Coordinate System.** Geometries, like text, can be positioned using either frame coordinates or grid coordinates. Grid coordinates are used by default. Unlike text, however, geometries are sized using the units of the coordinate system in which they are placed. That is, geometries that use frame coordinates use frame units to specify both the position and the size, while geometries that use grid coordinates use grid units to specify both the position and size. When frame coordinates are used, the geometry is locked at a particular location within the frame, and actions which modify the view (such as zooming, translating, and rotating) have no effect on the geometry. When grid coordinates are used, the geometry is part of the view, subject to change when the view changes. For example, a circle in frame coordinates retains its size and shape when you zoom into the plot, while a circle in grid coordinates grows as you zoom in. You specify the coordinate system using the Geometry dialog.

To specify a geometry's coordinate system:

1. In the workspace, select the geometry for which you want to specify a coordinate system.
2. On the sidebar, click Object Details. The Geometry dialog appears.
3. Select one of the two option buttons Frame or Grid in the region labeled Coordinate System.
4. Click Close.

**18.2.2.5. Controlling Geometry Position.** You specify the anchor position for a new geometry by clicking anywhere in the frame. You have some guidance in this by using the workspace rulers, and you can gain some specific control by using the Snap to Paper or Snap to Grid sidebar options. Users can also move a geometry to the front or the back of a group of geometries by selecting it, then choosing Edit, Push or Edit, Pop. However, for complete

control over the position of your anchor point, you can specify exact coordinates for the anchor position, or origin of the geometry, using the Origin controls in the Geometry dialog.

To specify an exact position for a geometry's origin:

1. In the workspace, select the geometry for which you want to specify an origin.
2. On the sidebar, click Object Details. The Geometry dialog appears.
3. Enter a value for the X-position in the text field labeled X, a value for the Y-position in the text field labeled Y, (and, for 3-D line geometries brought in from data files, a value for the Z-position in the text field labeled Z). Values must be in the coordinate system specified for the geometry, either frame units or grid units. Units other than Paper Ruler may be specified by typing them after the number. Use cm for centimeters, in for inches or pix for pixels.

**18.2.2.6. Specifying Geometry Scope.** By default, a geometry is displayed only in the frame in which it is created. You can, however, choose to have the geometry appear in all frames using the same data file as the one in which the geometry was created. Such frames are called like frames.

To propagate a geometry to all like frames:

1. In the workspace, select the geometry that you want to appear in like frames.
2. On the sidebar, click Object Details. The Geometry dialog appears.
3. Select the check box labeled Show in All Like Frames.
4. Click Close.

**18.2.2.7. Attaching Geometries to Zones or XY-Mappings.** By default, geometries are always displayed, regardless of which zones or XY-mappings are currently active. Sometimes, however, you draw a geometry to highlight a particular feature of a specific zone or XY-mapping, and that geometry is meaningless unless the zone or XY-map is displayed as well. In such cases, you can control the display of the geometry by attaching the geometry to the zone or XY-map.

To attach a geometry to a zone or XY-map:

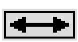
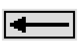
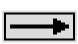
1. In the workspace, select the geometry that you want to attach to a zone or XY-map.
2. On the sidebar, click Object Details. The Geometry dialog appears.
3. Select the check box labeled Attach to Zone/Map.
4. Enter a zone or XY-mapping number in the text field immediately to the right of the Attach to Zone/Map check box.
5. Click Close.

Once attached to a zone or XY-map, the geometry is drawn when the zone or XY-map is active, and not drawn when the zone or XY-map is inactive.

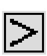
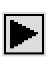
**18.2.2.8. Adding Arrowheads to Polylines.** You can add arrowheads to either or both ends of any polyline, and control the arrowhead style, size, and angle. You control polyline arrowheads from either the Geometry dialog or the Quick Edit dialog.

To control polyline arrowheads:

1. In the workspace, select the polyline or polylines for which you want to specify arrowhead attachment.
2. On the sidebar, click Object Details. If you have selected a single polyline, the Geometry dialog appears. Continue with Step 3. If you have selected multiple polylines, the Quick Edit dialog appears. Continue with Step 4.
3. (Geometry dialog) Specify where to place arrowheads for the geometry by selecting none, one, or both of the check boxes labeled Start and End beside the label Attachment. If you select one or both of Start and End, specify a style by selecting one of the option buttons labeled Plain, Filled, or Hollow. Specify a size by either choosing a pre-set value or entering a value in the Size (%) field. Specify an angle by either choosing a pre-set value or entering a value in the Angle (deg) field.
4. (Quick Edit dialog) Specify where to place arrowheads by clicking on one of the following:

   -  No arrowheads.
   -  Arrowheads at both ends.
   -  Arrowhead at start of polyline.
   -  Arrowhead at end of polyline.

   Specify a style by clicking on one of the following buttons:

   -  Plain arrowhead style.
   -  Filled arrowhead style.
   -  Hollow arrowhead style.

   Specify a size for the arrowheads by clicking Size, located beneath the Arrowhead Style options, then choosing either one of the pre-set sizes, or Enter. If you choose Enter, an Enter Value dialog appears. Enter the desired size as a percentage of the frame height, then click OK.

Specify an angle for the arrowheads by clicking Angle, then choosing either one of the preset values, or Enter. If you choose Enter, an Enter Value dialog appears. Enter the desired arrowhead angle, which is the angle that one side of the arrowhead makes with the polyline, in degrees.

**18.2.2.9. Specifying Circle Attributes.** Using the Geometry dialog, you can modify the radius of a circle and also the number of line segments used to approximate the circle.

To modify a circle's radius:

1. In the workspace, select the circle for which you want to modify the radius.
2. On the sidebar, click Object Details. The Geometry dialog appears.
3. Enter a new value in the text field labeled Radius. Units other than Paper Ruler may be specified by typing them after the number. Use cm for centimeters, in for inches or pix for pixels.
4. Click Close.

To modify the number of line segments used to approximate the circle:

1. In the workspace, select the circle for which you want to specify a different number of line segments.
2. On the sidebar, click Object Details. The Geometry dialog appears.
3. Enter a new value in the text field labeled Approximated by Number of Sides.
4. Click Close.

**18.2.2.10. Specifying Ellipse Attributes.** Using the Geometry dialog, you can modify the horizontal and vertical axes of an ellipse and also the number of line segments used to approximate the ellipse.

To modify an ellipse's axes:

1. In the workspace, select the ellipse for which you want to modify the axes.
2. On the sidebar, click Object Details. The Geometry dialog appears.
3. Enter a new value for the horizontal axis length in the text field labeled Horizontal Axis. Units other than Paper Ruler may be specified by typing them after the number. Use cm for centimeters, in for inches or pix for pixels.
4. Enter a new value for the vertical axis length in the text field labeled Vertical Axis. Units other than Paper Ruler may be specified by typing them after the number. Use cm for centimeters, in for inches or pix for pixels.
5. Click Close.

To modify the number of line segments used to approximate the ellipse:

1. In the workspace, select the ellipse for which you want to specify a different number of line segments.
2. On the sidebar, click Object Details. The Geometry dialog appears.
3. Enter a new value in the text field labeled Approximated by Number of Sides.
4. Click Close.

**18.2.2.11. Specifying a Square's Size.** The only specific attribute of a square that you can modify is the side length.

To modify the size of a square:

1. In the workspace, select the square for which you want to modify the size.
2. On the sidebar, click Object Details. The Geometry dialog appears.
3. Enter a new value in the text field labeled Size. Units other than Paper Ruler may be specified by typing them after the number. Use cm for centimeters, in for inches or pix for pixels.
4. Click Close.

**18.2.2.12. Specifying a Rectangle's Size.** The only specific attributes of a rectangle that you can modify are the width and height.

To modify the size of a rectangle:

1. In the workspace, select the rectangle for which you want to modify the size.
2. On the sidebar, click Object Details. The Geometry dialog appears.
3. Enter a new value in the text field labeled Width. Units other than Paper Ruler may be specified by typing them after the number. Use cm for centimeters, in for inches or pix for pixels.
4. Enter a new value in the text field labeled Height. Units other than Paper Ruler may be specified by typing them after the number. Use cm for centimeters, in for inches or pix for pixels.
5. Click Close.

**18.2.2.13. Moving Individual Points of a Geometry.** With the Adjustor tool, you can move one or more points from polyline geometries, as follows:

1. On the sidebar, choose the Adjustor tool by clicking .
2. Click the point you want to move. If you want to move more than one point, you can drag a box around them to select them as a group, or Shift-click each point in turn. Handles appear on the selected points. (The points do not all have to be in the same polyline.)
3. Drag the selected points to move them. The points move as a group.

As you move the points, you can use the V and H keys on your keyboard to restrict motion to the vertical and horizontal directions, respectively. Press A to allow movement in all directions (the default).

### 18.2.3. Creating 3-D Line Geometries

Three-dimensional line geometries cannot be created interactively; they must be created in a data file. For example, the following data file includes two 3-D line geometries representing the trajectories of two bugs in a cubic room (10 x 10 x 10). Two zones are used to represent the ceiling and floor of the room. (In order to display 3-D geometries, you must either include at least one zone in the data file with the 3-D geometries or read the 3-D geometries in, using the Add to Current Data Set option, after having first read a data set into the frame.) A plot of this data set (included in your Tecplot distribution as **`examples/dat/3dgeom.dat`**) is shown in Figure 18-8.

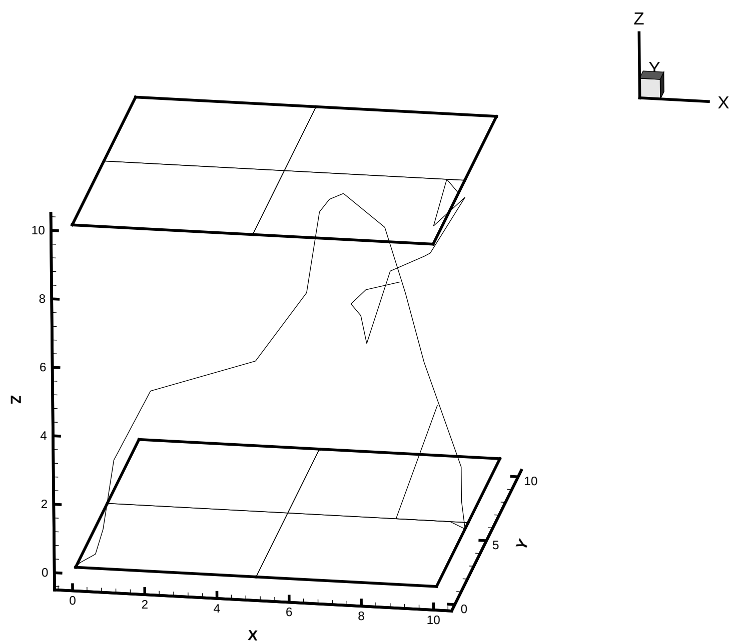



**Figure 18-8.** Three-dimensional line geometries.

```
TITLE = "EXAMPLE: 3D GEOMETRIES"
 VARIABLES = "X", "Y", "Z"
```

```
ZONE T="Floor", I=3, J=3, F=POINT
0. 0. 0.
5. 0. 0.
10. 0. 0.
0. 5. 0.
5. 5. 0.
10. 5. 0.
0. 10. 0.
5. 10. 0.
10. 10. 0.
ZONE T="Ceiling", I=3, J=3, F=POINT
0. 0. 10.
5. 0. 10.
10. 0. 10.
0. 5. 10.
5. 5. 10.
10. 5. 10.
0. 10. 10.
5. 10. 10.
10. 10. 10.
GEOMETRY X=0, Y=0, Z=0, CS=GRID, C=BLUE, T=LINE3D, F=POINT
1
12
10. 4.0 10.
9.5 5.0 10.0
9.3 4.0 9.0
10. 5.0 9.5
9.2 4.0 8.2
8.8 5.3 7.6
7.8 5.6 7.0
7.2 5.2 5.0
6.9 6.0 5.5
6.9 4.5 6.4
7.3 4.6 6.8
8.2 4.8 7.0
GEOMETRY X=0, Y=0, Z=0, CS=GRID, C=RED, T=LINE3D, F=BLOCK
1
20
0 .1 .5 .7 1. 2. 4.9 6.3 6.5 6.6 6.9 8. 8.5 9. 10. 10. 10. 9.5 8. 9.
0 .1 .3 .4 .5 .7 .8 1. 2. 3. 3.5 3.7 4. 4. 4. 4. 4.5 5. 5. 6.
0 .1 .3 1. 3. 5. 6. 8. 10. 10. 10. 9. 7. 5. 2. 1. 0. 0. 0. 3.
```

## 18.3. Pushing and Popping Text and Geometries

You can place text and geometries in any order you like. Tecplot draws all geometries first, in the order in which they were placed, then all text. There are times, however, when you want to override this default order. Sometimes, for example, you may place one geometry, then draw a second geometry that partially obscures the first. You can “pop” the first geometry to the top of the draw stack, so that Tecplot draws it after the geometry that had partially obscured it.

Popping raises the object to the top of its draw stack. Pushing lowers the object to the bottom of its draw stack. The text stack is always drawn on top of the geometry stack.

To push a text or geometry object:

1. Select the object.
2. From the Edit menu, choose Push.

To pop a text or geometry object:

1. Select the object.
2. From the Edit menu, choose Pop.

## 18.4. Aligning Text and Geometries

When you have a number of text and geometries, you may want to align them after placing them. You can do this using the alignment tools in the Quick Edit dialog, shown in Figure 18-9.

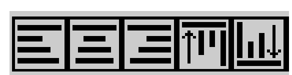


**Figure 18-9.** Alignment tools.

You can use these tools as follows:

1. On the sidebar, choose the Selector tool by clicking .
2. In the workspace, select a text or geometry with which you want to align other objects.
3. Drag the mouse to draw a rubber band box around the text and geometries you want to align. The Group Select dialog appears.
4. Select the Text and Geometries check boxes in the Group Select dialog, then click OK. Selection handles appear on the selected text and geometries.
5. On the sidebar, click Quick Edit to call up the Quick Edit dialog, if it is not already displayed.
6. Use the alignment buttons as follows:

   -  Left align the selected text and geometries with the original selected object.

-  Center the selected text and geometries with the original selected object.
-  Right align the selected text and geometries with the original selected object.
-  Top align the selected text and geometries with the original selected object.
-  Bottom align the selected text and geometries with the original selected object.

## 18.5. Linking Text and Geometries to Macros

Each text or geometry you create can be linked to a macro function. This macro function is called whenever the user holds down the control key and clicks the right mouse button on the text or geometry.

For example, if you have pieces of text, each representing a different well, Ctrl-right-click on any piece could run a macro that brings up an XY-plot of that well's data.

Macro functions are specified with the "Macro Function" field in the Geometry dialog or in the Text Options dialog. If desired, the macro function may be listed with one or more parameters. See Chapter 28, "Using Macros," and the *Tecplot Reference Manual*, for more detailed information on using macros in Tecplot.

## 18.6. Creating Custom Characters

You can create symbols, characters, and even custom fonts for use in Tecplot. See Section 31.5, "Defining Custom Characters and Symbols," for instructions.

# CHAPTER 19 Frame Linking

Tecplot's frame linking feature allows you to link specific style attributes between frames. Changing an attribute in one frame results in the same change to all other frames linked with respect to that attribute.

## 19.1. Attributes that can be Linked

Figure 19-1 shows the Set Links for Current Frame dialog.

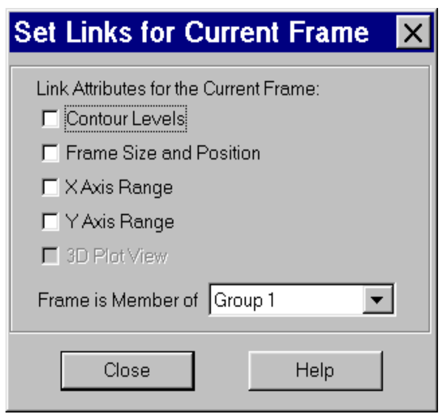



**Figure 19-1.** The Set Links for Current Frame dialog.

The attributes which may be linked are:

- **Contour Levels:** Link the values and number of contour levels.
- **Frame Size and Position:** Allows you to stack transparent frames. (See section 2.1.4, "Modifying the Frame Background Color," for further information.)
- **X-Axis Range:** For XY- and 2-D plots.
- **Y-Axis Range:** For XY- and 2-D plots.

- **3D Plot View:** Links the ranges of axes in 3D frame mode.

## 19.2. Frame Linking Groups

In addition to setting which attributes to link between frames, you can also choose to which group the current frame belongs. The first step when linking attributes between frames is to identify which frames are going to be assigned to which group. If you only have two frames, then this is unimportant as both frames default to being in group 1 and there is no need to have any other groups. Attributes are only propagated to other frames that are members of the same group and that have the same link attributes selected.

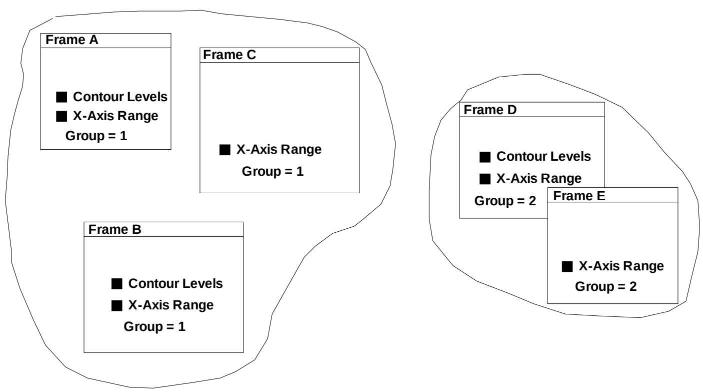



**Figure 19-2.** Five frames in two groups showing different linking options.

Figure 19-2 shows five frames. Frames A, B, and C are in group 1. Frames D and E are in group 2. Note that all five frames have their X-axis ranges linked. If you change the X-axis range in frame A, the corresponding change will occur in frames B and C. It will not occur in frames D and E, as they are not in group 1. A change to the X-axis range in frame D will only be propagated to frame E.

Some frames in Figure 19-2 also have linked contour levels. Changes in the contour levels in frame A would propagate to frame C only. Setting the link for contour levels in frame D has no effect as frame E, the only other frame in group 2, does not have the same attribute linked.

New frames added to a group take on the characteristics of previous members of the group. For this reason, it is important to start the group with a frame that has the characteristics you want for that group (though, once frames are in a group, a change to a linked attribute of one frame changes that attribute for all frames in that group).

## 19.3. Linking an Attribute

To link an attribute:

1. From the Frame menu, choose the Set Links for Current Frame option. The Set Links for Current Frame dialog appears.
2. Set the group number for this frame. To change the group the current frame is assigned to simply change the group setting at the bottom of the dialog.
3. Activate the check box for the attribute you wish to link.
4. One by one, pop the other frames you want to also have the same link attributes. Repeat Step 1-3 for each of these frames.

## 19.4. Dependent Axes

When 2-D or XY-frames have dependent axes and the axis ranges are linked, Tecplot will make a “best-fit” attempt to match the axis ranges between frames. Misalignments can occur when the aspect ratios for the lengths of the axes is not the same between two frames with linked X- and Y-axes. Setting the X- and Y-axes to be independent will allow a precise match.

CHAPTER 20

# Working with Finite-Element Data

A finite-element zone consists of a set of points (nodes) that are connected into polygonal or polyhedral units called elements. Associated with each element is a list of the nodes used by that element, in the order in which they are connected. A complete finite-element zone consists of the set of nodes plus the connectivity list for each element.

Tecplot supports both triangular and quadrilateral elements; these are collectively referred to as finite-element surface zones. Tecplot also supports tetrahedral or brick polyhedral elements, which are referred to as finite-element volume zones. Each Tecplot zone must be composed exclusively of one element type. You can, however, simulate zones with mixed element types by repeating nodes as necessary. Thus, a triangle element can be included in a quadrilateral zone by repeating one node in the element's connectivity list, and tetrahedral, pyramidal, and prismatic elements can be included in a brick zone by repeating nodes appropriately.

While finite-element data is usually associated with numerical analysis for modeling complex problems in 3-D structures, heat transfer, fluid dynamics, and electromagnetics, it also provides an effective approach for organizing data points in or around complex geometrical shapes. For example, you may not have the same number of data points on different lines, there may be holes in the middle of the data set, or the data points may be irregularly (randomly) positioned. For such difficult cases, you may be able to organize your data as a patchwork of elements. Each element can be virtually independent of the other elements, so you can group your elements to fit complex boundaries and leave voids within sets of elements. Figure 20-1 shows how finite-element data can be used to model a complex boundary.

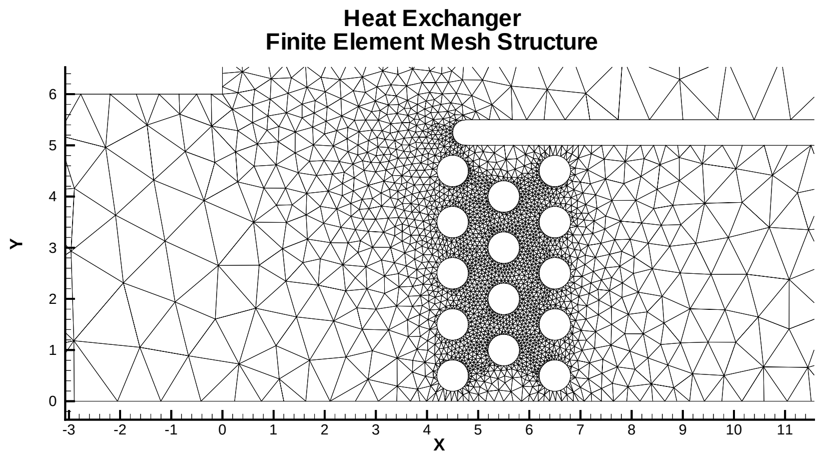



**Figure 20-1.** Finite-element data used to model a complex boundary.

## 20.1. Creating Finite-Element Data Sets

Creating a finite-element data set is generally somewhat more work than creating a similar sized ordered data set, because in addition to specifying all the data points, you must also specify the connectivity list which describes how the data points are connected into elements. As an example, consider the data shown in Table 20-1.

| Node | X | Y | P | T |
|---|---|---|---|---|
| A | 0.0 | 1.0 | 100.0 | 1.6 |
| B | 1.0 | 1.0 | 150.0 | 1.5 |
| C | 3.0 | 1.0 | 300.0 | 2.0 |
| D | 0.0 | 0.0 | 50.0 | 1.0 |
| E | 1.0 | 0.0 | 100.0 | 1.4 |
| F | 3.0 | 0.0 | 200.0 | 2.2 |
| G | 4.0 | 0.0 | 400.0 | 3.0 |
| H | 2.0 | 2.0 | 280.0 | 1.9 |

**Table 20-1.** Finite-element data.

We can create an **FEPOINT** Tecplot data file for this data set as follows (a 2-D mesh plot of this data set is shown in Figure 20-2):

```
TITLE = "Example: 2D Finite-Element Data"
VARIABLES = "X", "Y", "P", "T"
ZONE N=8, E=4, F=FEPOINT, ET=QUADRILATERAL
0.0 1.0 100.0 1.6
1.0 1.0 150.0 1.5
3.0 1.0 300.0 2.0
0.0 0.0 50.0 1.0
1.0 0.0 100.0 1.4
3.0 0.0 200.0 2.2
4.0 0.0 400.0 3.0
2.0 2.0 280.0 1.9
1 2 5 4
2 3 6 5
6 7 3 3
3 2 8 8
```

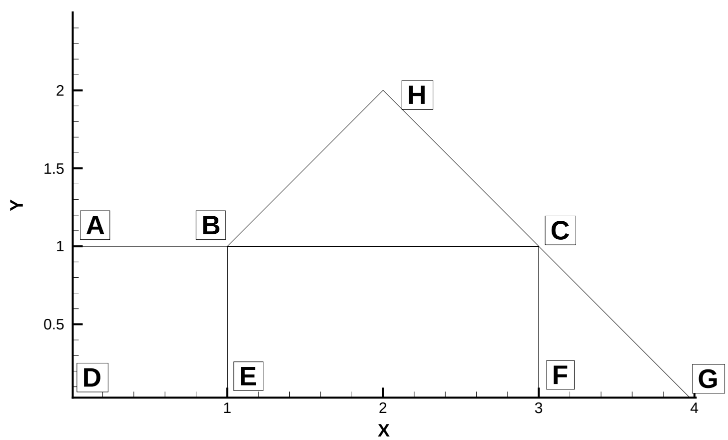



**Figure 20-2.** A mesh plot of 2-D finite-element data.

The **ZONE** record describes completely the form and format of the data set: there are eight nodes, indicated by the parameter **N=8**; four elements, indicated by the parameter **E=4**, and the elements are all quadrilaterals, as indicated by the parameter **ET=QUADRILATERAL**.

The same data file can be written more compactly in **FEBLOCK** format as follows:

```
TITLE = "Example: 2D Finite-Element Data"
VARIABLES = "X", "Y", "P", "T"
ZONE N=8, E=4, F=FEBLOCK, ET=QUADRILATERAL
0.0 1.0 3.0 0.0 1.0 3.0 4.0 2.0
1.0 1.0 1.0 0.0 0.0 0.0 0.0 2.0
100.0 150.0 300.0 50.0 100.0 200.0 400.0 280.0
1.6 1.5 2.0 1.0 1.4 2.2 3.0 1.9
1 2 5 4
2 3 6 5
6 7 3 3
3 2 8 8
```

In **FEBLOCK** format, all values for a single variable are written in a single block. The length of the block is the number of data points in the zone. In **FEPOINT** format, all variables for a single data point are written in a block, with the length of the block equal to the number of variables. The connectivity list, however, is the same for both formats.

You can change the connectivity list to obtain a different mesh for the same data points. In the above example, substituting the following connectivity list yields the five-element mesh shown in Figure 20-3. (You must also change the **E** parameter in the zone control line to specify five elements.)

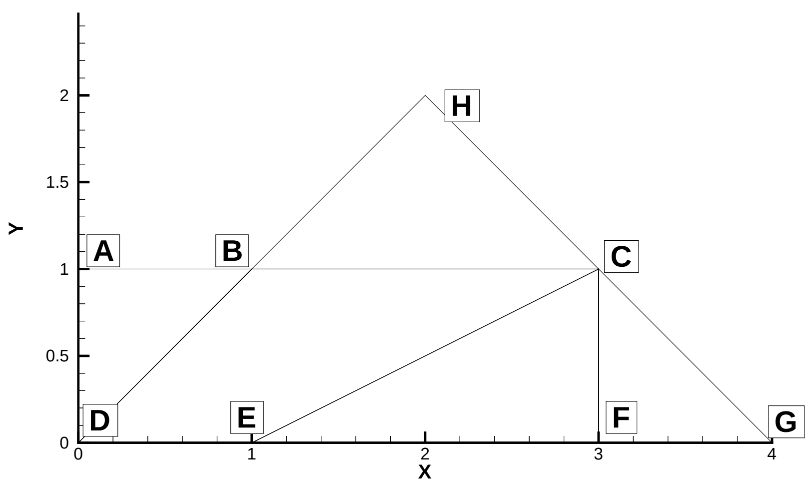



**Figure 20-3.** Finite-element data of Figure 20-2 with different connectivity list.

```
1 2 4 4
4 2 3 5
5 3 6 6
6 7 3 3
3 2 8 8
```

Finite-element surface data specify node locations in three dimensions. For example, consider the data in Table 20-2. Locations are listed for eleven nodes, each having only the three spatial variables X, Y, and Z. We would like to create an finite-element surface zone with this data set, where some of the elements are triangles and some are quadrilaterals. All the elements could be organized into one zone, of element type Quadrilateral, but as an illustration of creating 3-D surface data, create three zones: one triangular, one quadrilateral, and one a mixture (using quadrilaterals with repeated nodes for the triangles).

| X | Y | Z |
|---|---|---|
| 0.0 | 0.0 | 1.0 |
| 0.0 | 0.0 | –2.0 |
| 1.0 | 0.0 | –2.0 |
| 1.0 | 1.0 | 0.0 |
| 1.0 | 1.0 | –1.0 |
| 1.0 | –1.0 | 0.0 |
| 1.0 | –1.0 | –1.0 |
| –1.0 | 1.0 | 0.0 |
| –1.0 | 1.0 | –1.0 |
| –1.0 | –1.0 | 0.0 |
| –1.0 | –1.0 | –1.0 |

**Table 20-2.** Data set with eleven nodes and three variables.

A Tecplot data file for the data in Table 20-2 is shown below in **`FEPOINT`** format and plotted in Figure 20-4:

```
TITLE = "Example: 3D FE-SURFACE ZONES"
VARIABLES = "X", "Y", "Z"
ZONE T="TRIANGLES", N=5, E=4, F=FEPOINT, ET=TRIANGLE
0.0 0.0 1.0
-1.0 -1.0 0.0
-1.0 1.0 0.0
1.0 1.0 0.0
1.0 -1.0 0.0
1 2 3
1 3 4
1 4 5
1 5 2
ZONE T="PURE-QUADS", N=8, E=4, F=FEPOINT, ET=QUADRILATERAL
-1.0 -1.0 0.0
-1.0 1.0 0.0
1.0 1.0 0.0
1.0 -1.0 0.0
-1.0 -1.0 -1.0
-1.0 1.0 -1.0
```

```
1.0 1.0 -1.0
1.0 -1.0 -1.0
1 5 6 2
2 6 7 3
3 7 8 4
4 8 5 1
ZONE T="MIXED", N=6, E=4, F=FEPOINT, ET=QUADRILATERAL
-1.0 -1.0 -1.0
-1.0 1.0 -1.0
1.0 1.0 -1.0
1.0 -1.0 -1.0
0.0 0.0 -2.0
1.0 0.0 -2.0
1 5 2 2
2 5 6 3
3 4 6 6
4 1 5 6
```

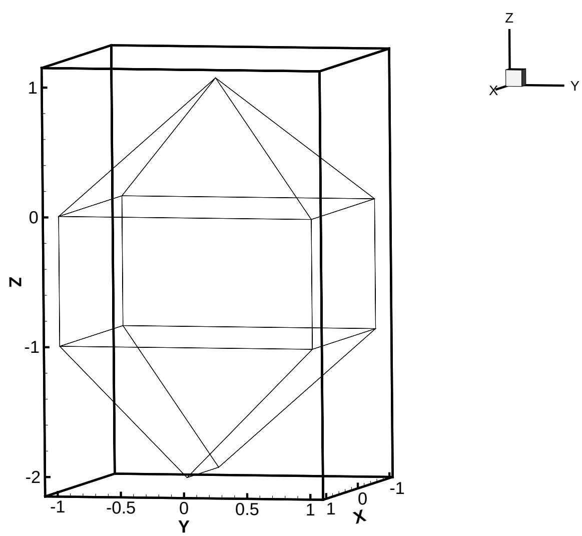



**Figure 20-4.** Three-dimensional mesh plot of finite-element surface data.

# 20.2. Creating 3-D Volume Data Files

Finite-element volume data in Tecplot is constructed from either tetrahedrons having four nodes or bricks having eight nodes. Bricks are more flexible, because they can be used (through the use of repeated nodes in the connectivity list) to construct elements with fewer than eight nodes and combine those elements with bricks in a single zone. Bricks, on the other hand, are harder to construct because care must be taken to make sure all faces in all bricks are planar.

## 20.2.1. Creating a Finite-Element Volume Brick Data Set

As a simple example of finite-element volume brick data, consider the data in Table 20-3. The data can be divided into five brick elements, each of which is defined by eight nodes.

| X | Y | Z | Temperature |
|---|---|---|---|
| 0.0 | 0.0 | 0.0 | 9.5 |
| 1.0 | 1.0 | 0.0 | 14.5 |
| 1.0 | 0.0 | 0.0 | 15.0 |
| 1.0 | 1.0 | 1.0 | 16.0 |
| 1.0 | 0.0 | 1.0 | 15.5 |
| 2.0 | 2.0 | 0.0 | 17.0 |
| 2.0 | 1.0 | 0.0 | 17.0 |
| 2.0 | 0.0 | 0.0 | 17.5 |
| 2.0 | 2.0 | 1.0 | 18.5 |
| 2.0 | 1.0 | 1.0 | 20.0 |
| 2.0 | 0.0 | 1.0 | 17.5 |
| 2.0 | 2.0 | 2.0 | 18.0 |
| 2.0 | 1.0 | 2.0 | 17.5 |
| 2.0 | 0.0 | 2.0 | 16.5 |

**Table 20-3.** Data with fourteen nodes and four variables.

In each element's connectivity list, Tecplot draws connections from each node to three other nodes. You can think of the first four nodes in the element as the "bottom" layer of the brick, and the second four nodes as the "top." Within the bottom or top layer, nodes are connected cyclically (1-2-3-4-1; 5-6-7-8-5); the layers are connected by connecting corresponding nodes (1-5; 2-6; 3-7; 4-8). Figure 20-5 illustrates this basic connectivity. When you are creating your own connectivity lists for brick elements, you must keep this basic connectivity in mind, par-

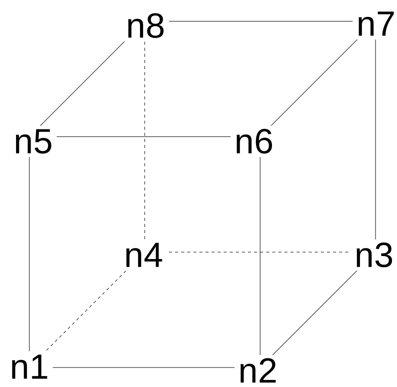



**Figure 20-5.** Basic connectivity for finite-element bricks.

ticularly when using duplicated nodes to create pyramids and wedges. Tecplot lets you create elements that violate this basic connectivity, but the result will probably not be what you want.

The data file in **FEPOINT** format is shown below:

```
TITLE = "Example: FE-Volume Brick Data"
VARIABLES = "X", "Y", "Z", "Temperature"
ZONE N=14, E=5, F=FEPOINT, ET=BRICK
0.0 0.0 0.0 9.5
1.0 1.0 0.0 14.5
1.0 0.0 0.0 15.0
1.0 1.0 1.0 16.0
1.0 0.0 1.0 15.5
2.0 2.0 0.0 17.0
2.0 1.0 0.0 17.0
2.0 0.0 0.0 17.5
2.0 2.0 1.0 18.5
2.0 1.0 1.0 20.0
2.0 0.0 1.0 17.5
2.0 2.0 2.0 18.0
2.0 1.0 2.0 17.5
2.0 0.0 2.0 16.5

1 1 1 1 2 4 5 3
2 4 5 3 7 10 11 8
4 4 5 5 10 13 14 11
```

```
4 4 4 4 9 12 13 10
2 2 4 4 7 6 9 10
```

The same data in **FEBLOCK** format is shown below:

```
TITLE = "Example: FE-Volume Brick Data"
VARIABLES = "X", "Y", "Z", "Temperature"
ZONE N=14, E=5, F=FEBLOCK, ET=BRICK
0.0 1.0 1.0 1.0 1.0 2.0 2.0 2.0 2.0 2.0 2.0 2.0 2.0 2.0
0.0 1.0 0.0 1.0 0.0 2.0 1.0 0.0 2.0 1.0 0.0 2.0 1.0 0.0
0.0 0.0 0.0 1.0 1.0 0.0 0.0 0.0 1.0 1.0 1.0 2.0 2.0 2.0
9.5 14.5 15.0 16.0 15.5 17.0 17.0
17.5 18.5 20.0 17.5 18.0 17.5 16.5

1 1 1 1 2 4 5 3
2 4 5 3 7 10 11 8
4 4 5 5 10 13 14 11
4 4 4 4 9 12 13 10
2 2 4 4 7 6 9 10
```

Figure 20-6 shows the resulting mesh plot from the data set listed in this section.

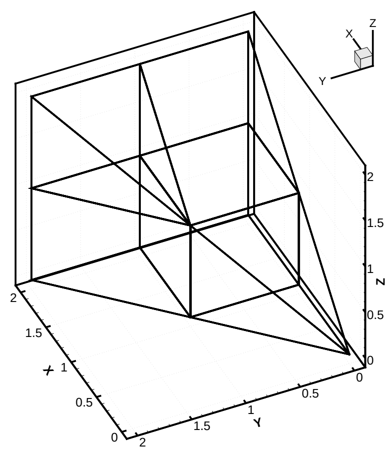



**Figure 20-6.** A finite-element brick zone.

## 20.2.2. Creating a Finite-Element Volume Tetrahedral Data Set

As a simple example of an finite-element volume data set using tetrahedral elements, consider the data in Table 20-4. The data set consists of thirteen nodes, with seven variables. The nodes are to be connected to form twenty tetrahedral elements, each with four nodes.

| X | Y | Z | C | U | V | W |
|---|---|---|---|---|---|---|
| 0 | 0 | –95 | –1 | 1 | 0 | 8 |
| 0 | 85 | –42 | 0 | –5 | –3 | 9 |
| 81 | 26 | –42 | 2 | –22 | 80 | 8 |
| 50 | –69 | –42 | –6 | 72 | 52 | 9 |
| –50 | –69 | –42 | 14 | 67 | –48 | 9 |
| –81 | 26 | –2 | 20 | –30 | –82 | 9 |
| 0 | 0 | 0 | 1 | –2 | –5 | 10 |
| 50 | 69 | 43 | 14 | –68 | 48 | 11 |
| 81 | –26 | 43 | 20 | 31 | 82 | 11 |
| 0 | –85 | 43 | 0 | 84 | –3 | 10 |
| –81 | –26 | 43 | 2 | 21 | –80 | 11 |
| –50 | 69 | 43 | –6 | –71 | –51 | 11 |
| 0 | 0 | 96 | 1 | 0 | –1 | 12 |

**Table 20-4.** Data with thirteen nodes and seven variables.

The data file in **`FEPOINT`** format for the data in Table 20-4 is shown below, and plotted in Figure 20-7:

```
TITLE = "Example: FE-Volume Tetrahedral Data"
VARIABLES = "X", "Y", "Z", "C", "U", "V", "W"
ZONE N=13, E=20, F=FEPOINT, ET=TETRAHEDRON
0 0 -95 -1 1 0 8
0 85 -42 0 -85 -3 9
81 26 -42 2 -22 80 8
50 -69 -42 -6 72 52 9
-50 -69 -42 14 67 -48 9
-81 26 -42 20 -30 -82 9
0  0 0 1 -2 -5 10
50 69 43 14 -68 48 11
81 -26 43 20 31 82 11
0 -85 43 0 84 3 10
-81 -26 43 2 21 -80 11
-50 69 43 -6 -71 -51 11
```

```
0  0 96 1 0 -1 12

1 2 3 7
1 3 4 7
1 4 5 7
1 5 6 7
1 6 2 7
2 8 3 7
3 9 4 7
4 10 5 7
5 11 6 7
6 12 2 7
12 2 8 7
8 3 9 7
9 4 10 7
10 5 11 7
11 6 12 7
12 8 13 7
8 9 13 7
9 10 13 7
10 11 13 7
11 12 13 7
```

This data file is included in your Tecplot distribution's **examples/data** directory as the file **fetetpt.dat**. A block format version of the same data is included as the file **fetetbk.dat**.

## 20.3. Triangulated Data Sets

One common source of finite-element surface data is Tecplot's triangulation option. If you have 2-D data without a mesh structure, it is probably simplest to enter your data points as an I-ordered data set, then use Tecplot's triangulation feature to create a finite-element data set. You can then edit the file, and particularly the connectivity list, to obtain the set of elements you want, rather than having to create the entire connectivity list by hand.

For example, consider again the data of Table 20-1. We can triangulate that data set as follows:

1. Enter the data as a simple ordered data file, as follows:

```
VARIABLES = "X", "Y", "P", "T"
0.0 1.0 100.0 1.6
1.0 1.0 150.0 1.5
3.0 1.0 300.0 2.0
0.0 0.0 50.0 1.0
```

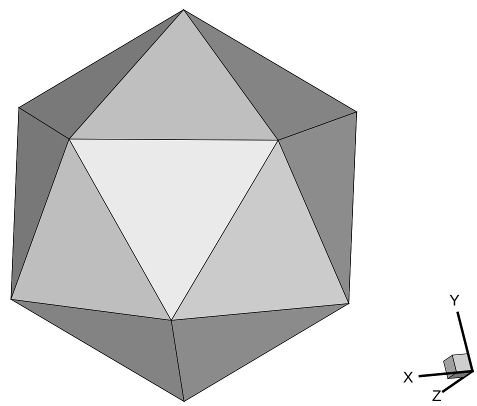



**Figure 20-7.** Finite-element volume tetrahedral data.

```
1.0 0.0 100.0 1.4
3.0 0.0 200.0 2.2
4.0 0.0 400.0 3.0
2.0 2.0 280.0 1.9
```

2. Read the data file into Tecplot and switch the frame mode to 2D.
3. From the Data menu, choose Triangulate, then select the simple ordered zone as the source zone, and click Compute.
4. From the File menu, choose Write Data File. The Write Data File Options dialog appears.
5. Select the ASCII check box and the Point Format check box, then click OK.
6. Save to a file name of your choice. The result is the following finite-element surface zone (in addition to your original zone):

```
VARIABLES = "X"
"Y"
"P"
"T"
```

```
ZONE T="Triangulation"
 N=8, E=7,F=FEPOINT ET=Triangle
DT=(SINGLE SINGLE SINGLE SINGLE )
 0.000000000E+000 1.000000000E+000 1.000000000E+002 1.600000024E+000
 1.000000000E+000 1.000000000E+000 1.500000000E+002 1.500000000E+000
 3.000000000E+000 1.000000000E+000 3.000000000E+002 2.000000000E+000
 0.000000000E+000 0.000000000E+000 5.000000000E+001 1.000000000E+000
 1.000000000E+000 0.000000000E+000 1.000000000E+002 1.399999976E+000
 3.000000000E+000 0.000000000E+000 2.000000000E+002 2.200000048E+000
 4.000000000E+000 0.000000000E+000 4.000000000E+002 3.000000000E+000
 2.000000000E+000 2.000000000E+000 2.800000000E+002 1.899999976E+000
 2 3 5
 5 4 2
 4 1 2
 7 6 3
 5 3 6
 3 2 8
 8 2 1
```

Figure 20-8 shows a plot of the resulting data. With triangulation, we obtain more elements (seven) than when we created the data set by hand (four), and the elements are triangles (naturally) rather than quadrilaterals. However, when you have many data points, triangulation is the most reasonable approach.

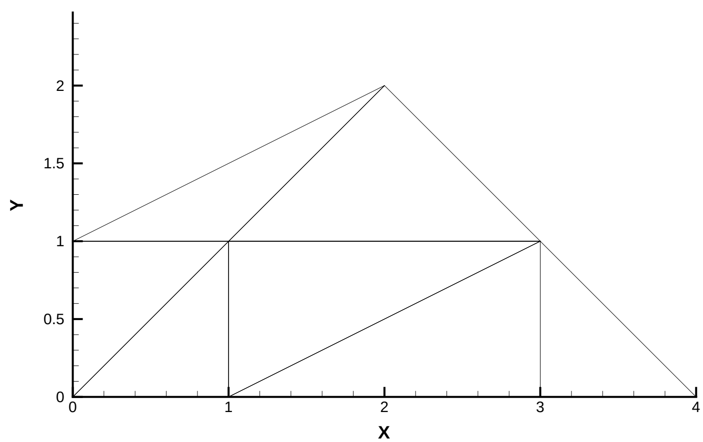



**Figure 20-8.** Triangulated data from Table 20-1.

## 20.4. Extracting Boundaries of Finite-Element Zones

Boundary lines for finite-element data are similar to boundary lines in ordered data, with a few exceptions. For triangular and quadrilateral meshes a boundary is drawn along the edges of elements that have no neighboring element according to the element connectivity. In some cases finite-element data will be supplied to Tecplot where each element is independent of all the others, that is, elements do not share common nodes. For this type of data a boundary will be drawn around each element.

Finite-element volume data, such as tetrahedral and brick element types, will not plot boundary lines, as opposed to ordered volume data. With finite-element volume data there are, by definition, no boundary lines. However, some plot styles will draw on the outer surface of these zones, in effect they are just drawing on the boundary. Extracting the boundary of these zones extracts the outer surface.

To extract the boundary of a finite-element zone:

1. Choose the appropriate frame mode for your data, either 2D or 3D. If the zone for which you are extracting the boundary is a 3-D surface, make sure the frame mode is set to 3D. If you create the boundary zone in 2D frame mode, the Z-coordinate is not taken into account, and points that are not coincident in 3D frame mode may become coincident in 2D mode. Tecplot eliminates coincident points in the final phase of the boundary extraction, so you could lose important boundary points.
2. From the Data menu, choose Extract, then choose FE-Boundary. The Extract FE-Boundary dialog appears as in Figure 20-9.

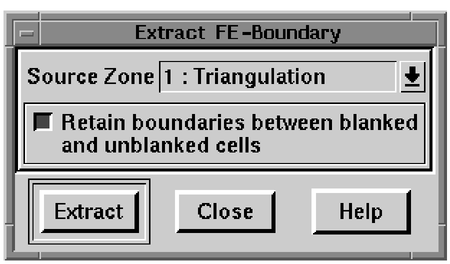



**Figure 20-9.** Extract FE-Boundary dialog.

3. Choose the source zone, that is, the zone for which you want to extract the boundary zone.
4. If blanking is on, decide whether to include the boundary between blanked and un-blanked cells in the zone boundary. To include this boundary, select the check box labeled Retain boundaries between blanked and un-blanked cells.

5. Click Extract. The extracted boundary zone is an FE-surface zone with quadrilateral elements, but each element has two repeated nodes so that each element is a single line segment along the boundary.

## 20.5. Limitations of Finite-Element Data

Working with finite-element data has some limitations, as follows:

- Finite-element data cannot be smoothed.
- Finite-element data cannot be mathematically differentiated.
- XY-plots of finite-element data treat the data as I-ordered; that is, the connectivity list is ignored. Only nodes are plotted, not elements, and the nodes are plotted in the order in which they appear in the data file.
- Index skipping in vector and scatter plots treats finite-element data as I-ordered; the connectivity list is ignored. Nodes are skipped according to their order in the data file.

# CHAPTER 21 Working with 3-D Volume Data

This chapter brings together descriptions of most of the Tecplot tasks involving 3-D volume data, whether IJK-ordered or finite-element. In this chapter, you will find descriptions of the following common 3-D volume tasks:

- Choosing which surfaces you want to plot from your volume data.
- Choosing which data points to use for vector and scatter plots.
- Interpolating 3-D volume irregular data.
- Extracting I-, J-, and K-planes from an IJK-ordered zone.
- Generating and extracting iso-surfaces.
- Extracting the outer surface of an FE-volume zone.
- Generating and extracting volumes with a plane.
- Creating specialized 3-D volume plots.

Other related topics such as IJK-blanking and animating IJK-planes are discussed in Chapter 25, “Blanking,” and Chapter 28, “Animation and Movies,” respectively.

## 21.1. Choosing Which Surfaces to Plot

There are many ways to divide volume data for plotting. One way to view volume data is to select surfaces from part of the data. For example, a typical plot would view a contour flooded plot drawn only on the outer surface of the volume data.

In Tecplot you may choose which surfaces to plot for volume zones from the Volume page of the Plot Attributes dialog. You can call up the Plot Attributes dialog by clicking Plot Attributes on the sidebar, or by double-clicking on a zone.

Figure 21-1 shows the Volume page of the Plot Attributes dialog.

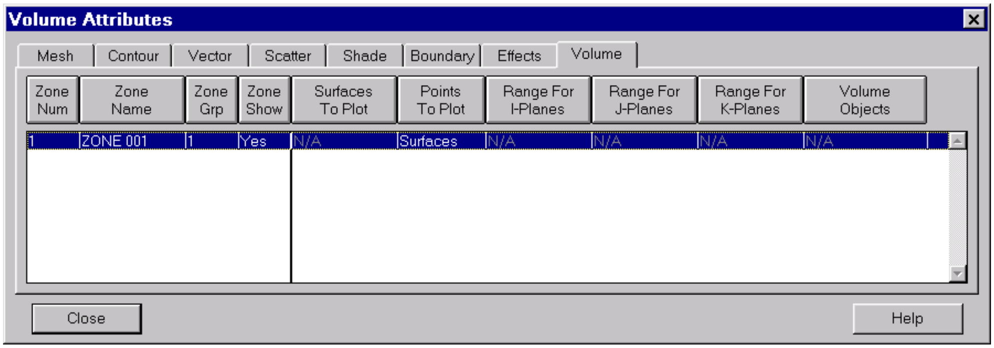



**Figure 21-1.** The Volume page of the Plot Attributes dialog.

The Surfaces to Plot option allows you to choose on of the following:

- **Boundary Cell Faces:** This will plot all surfaces on the outside of the volume zone. For IJK-ordered data this amounts to plotting the minimum and maximum I-, J-, and K-planes. For finite-element volume data this will plot all faces that do not have a neighbor cell (according to the connectivity list). If blanking is turned on, the boundary cells in the blanked region will not be drawn and you will be able to see the interior of the volume zone. Figure 21-2 shows plots of a volume zone with surface to plot set to Boundary Cell Faces without blanking, with value blanking, and with IJK-blanking.
- **Exposed Cell Faces (Default):** This setting is similar to the Boundary Cell Faces setting, save for cases in which value blanking is turned on. When value blanking is used the outer surfaces are drawn, similar to results from the Boundary Cell Faces setting. In addition, the cells faces between blanked and non-blanked cells are drawn. Figure 21-3 shows a plot of a volume zone with Surfaces to Plot set to Exposed Cell Faces with and without value blanking.
- **Planes Settings (I-, J-, K-, IJ-, JK-, IK-, and IJK-planes):** These settings will plot the appropriate combination of I-, J-, and or K-planes. The planes are determined by the Range for columns to the right of the dialog. These settings are available only for IJK-ordered data. Figure 21-4 shows a number of examples of plotting I-, J-, and K-planes.
- **Every Surface (Exhaustive):** This setting will plot every face of every cell in volume data. It is not recommended for large data sets. Unless the surfaces are translucent, the plot will appear the same as for the Exposed Cell Faces setting.

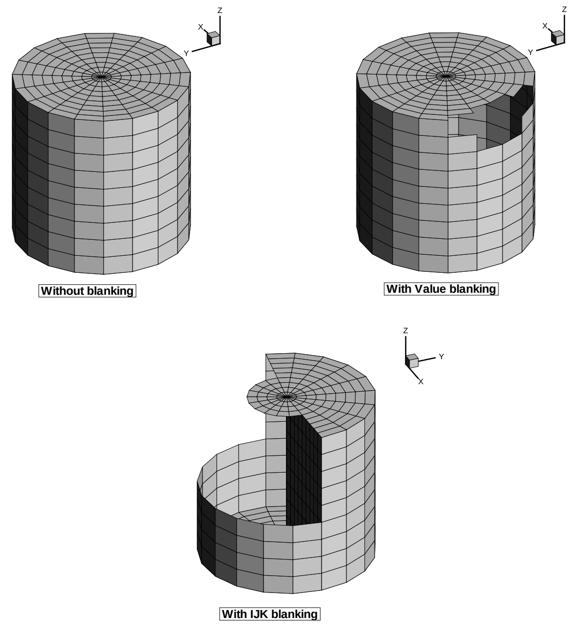



**Figure 21-2.** Boundary Cell Face plotting without blanking, with value-blanking, and with IJK-blanking.

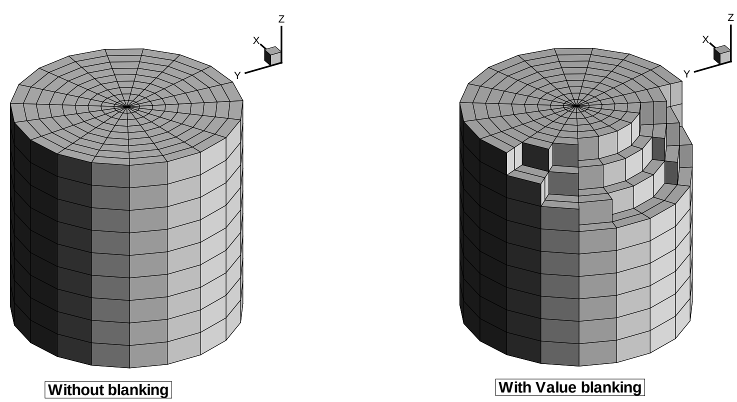



**Figure 21-3.** Examples of plots where Surfaces to Plot has been set to Exposed Cell Faces with (left) and without (right) value-blanking.

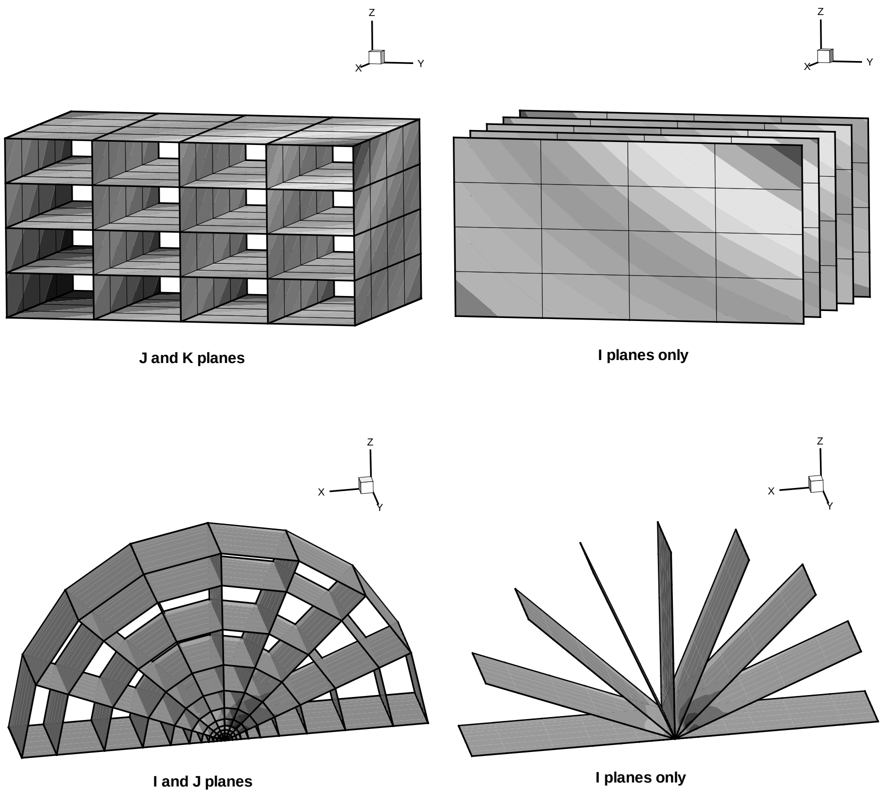



**Figure 21-4.** Examples of plotting I-, J-, and K-planes.

## 21.2. Choosing which Points to Plot

You may select the source for the data points used to plot vectors and scatter symbols from the Points to Plot column on the Plot Attribute dialog's Volume page. Your choices are Surfaces Only and All.

Choosing Surfaces Only will draw a vector or scatter symbol (when the appropriate zone layer is active) at every data point on all surfaces being plotted. To select the surface use the Surfaces to Plot option, discussed in the section above.

Choosing All will enable the plotting of vector or scatter symbols at every data point.

A plot where zone 1 is plotting scatter symbols only on one plane (*J=5*) and zone 2 is plotting all symbols is shown in Figure 21-5.

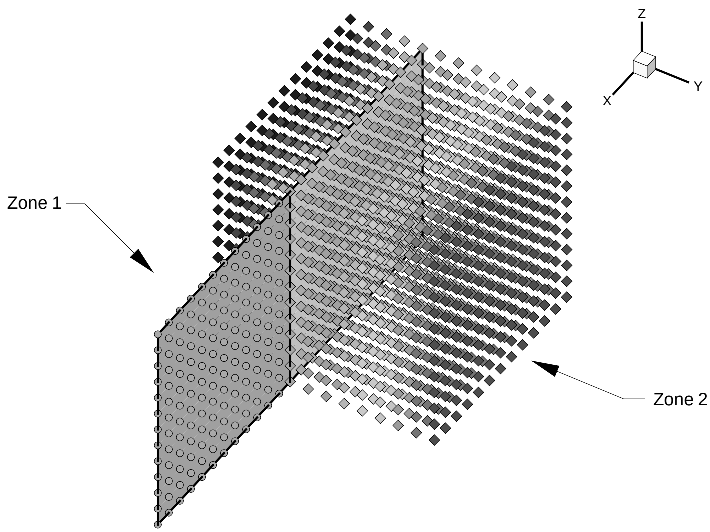



**Figure 21-5.** A plot showing two zones set to show only J-planes equal to five, with scatter symbols plotted on the surface in zone 1 and all symbols in zone 2.

In addition to selecting which surfaces to use to plot vector and scatter symbols, you may further limit these objects by setting the scatter symbol skip or vector skip on their respective pages of the Plot Attributes dialog. You can only set a skip value for ordered zones.

## 21.3. Plotting Derived Volume Objects

Volume streamlines, volume streamribbons, volume streamrods, slices and iso-surfaces are all derived from volume data automatically. The data used to generate these objects only exists for the life of the frame they are plotted in. When you save a layout or the style of a frame only the instructions necessary to recreate these objects are saved.

From the Volume Objects column on the Plot Attributes dialog you may include or exclude volume zones from consideration in the construction of volume objects. Figure 21-6 shows a plot with two zones where streamribbons and an iso-surface have been excluded from zone 2.

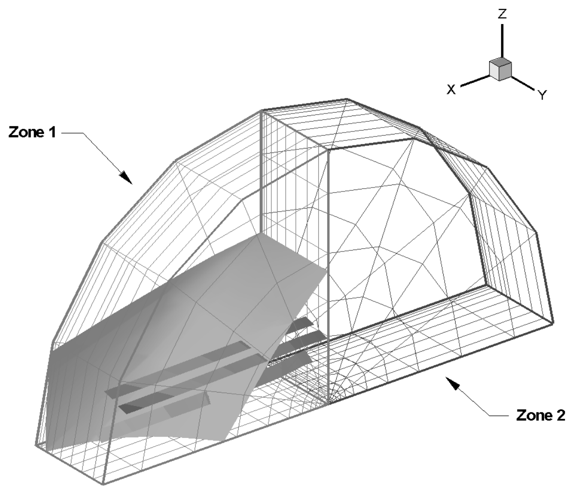



**Figure 21-6.** A plot where streamribbons and an iso-surface have been excluded from zone 2.

## 21.4. Interpolating 3-D Volume Irregular Data

To use 3-D volume irregular data in Tecplot field plots, you must interpolate the data onto a regular, IJK-ordered zone. (Tecplot does not have a 3-D equivalent for triangulation.) To interpolate your data, perform the following steps:

1. Place your 3-D volume irregular data into an I-ordered zone in a data file.
2. Read in your data file and create a 3-D scatter plot.
3. From the Data menu, choose Create Zone, then choose Rectangular. The Create Rectangular Zone dialog appears.
4. Enter the I-, J-, and K-dimensions for the new zone; at a minimum, you should enter 10 for each dimension. The higher the dimensions, the finer the interpolation grid, but the longer the interpolating and plotting time.
5. Enter the minimum and maximum X, Y, and Z values for the new zone. The default values are the minimums and maximums of the current (irregular) data set.
6. Click Create to create the new zone, and close to dismiss the dialog.
7. From the Data menu, choose Interpolate, then choose Kriging. The Kriging dialog appears (alternatively, choose Inverse Distance).

8. Choose the irregular data zone as the source zone, and the newly created IJK-ordered zone as the destination zone. Set any other kriging parameters as desired (see Section 25.9.2, “Kriging,” for details).
9. Click Compute to perform the kriging.

Once Tecplot completes the interpolation, you can plot the new IJK-ordered zone as any other 3-D volume zone. You may plot iso-surfaces, volume streamtraces, and so forth. At this point, you may want to deactivate or delete the original irregular zone so as not to conflict with plots of the new zone.

Figure 21-7 shows an example of irregular data interpolated into an IJK-ordered zone, with iso-surfaces plotted on the resultant zone.

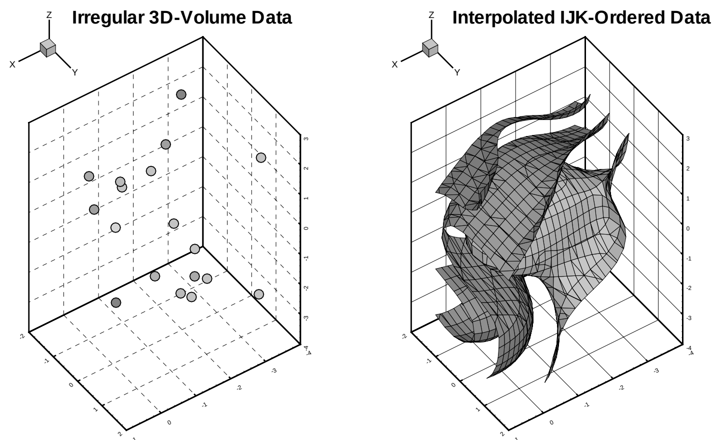



**Figure 21-7.** Irregular data interpolated into an IJK-ordered zone.

## 21.5. Extracting I-, J-, and K-Planes

Suppose you want to plot a collection of I-, J-, and K-planes that cannot be specified using the index range and index skip options of the Volume Attributes dialog. You can plot an arbitrary set of planes in Tecplot, but you must first extract each plane as a separate zone. Extracting planes is very simple using the Create SubZone dialog.

To extract a K-plane from an IJK-ordered zone, follow these steps:

1. From the Data menu, choose Create Zone, then choose SubZone. The Create SubZone dialog appears as shown in Figure 21-8.

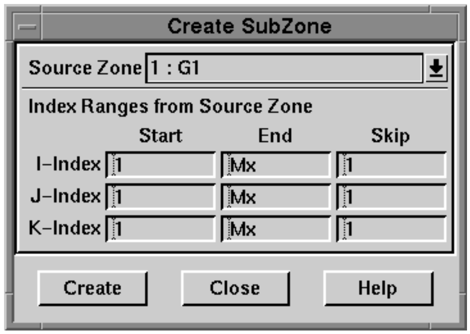



**Figure 21-8.** The Create SubZone dialog.

2. From the Source Zone drop-down, select the IJK-ordered zone.
3. In the K-Index fields, set Start and End to the same value: the number of the desired K-plane. Set Skip to 1.

   For example, to extract the *K=5* plane, set Start to 5, End to 5, and Skip to 1.
4. Click Create to extract the plane. Tecplot creates an IJ-ordered zone containing just the data points in the extracted plane.

Extracting I- and J-planes is similar.

# 21.6. Generating and Extracting Iso-Surfaces

An iso-surface is a surface having a constant value for the contour variable. Iso-surfaces require that your data contains volume zones, such as IJK-ordered, finite-element brick, or finite-element tetrahedral zones. In Tecplot you control iso-surfaces from the 3D Iso-Surface Details dialog under the Field menu, shown in Figure 21-9.

## 21.6.1. Locating Iso-Surfaces

The contour value where iso-surfaces are defined can either be associated with the current set of contour levels, or you may specify up to three unique levels independent of the contour levels. To enter unique levels, chose the 1, 2, or 3 Specified Values option from the 3D Iso-

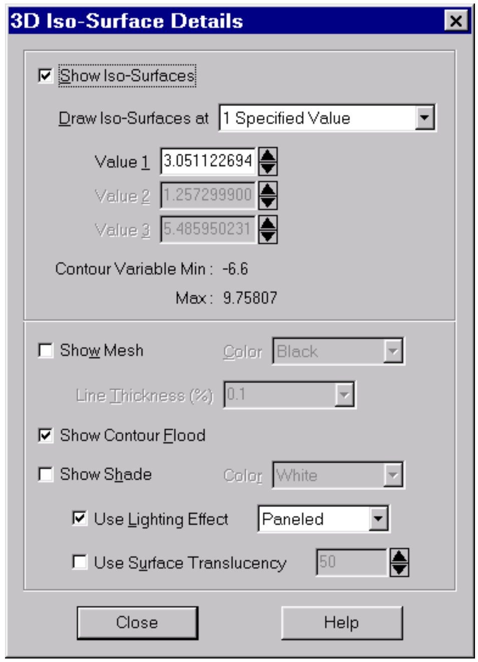



**Figure 21-9.** The 3D Iso-Surface Details dialog.

Surface Details dialog. You can then either enter values in the appropriate text fields, or use the increase or decrease arrows.

If you choose the Each Contour Level option to draw iso-surfaces, you may control the iso-surface positions using the Contour Levels dialog from the Field menu, or by using a contour level tool from the sidebar. The Contour Add and Contour Delete tools may also be used to add or delete iso-surfaces because they add and delete contour levels. The Contour Add tool may also adjust existing levels if you hold down the Ctrl key while clicking and dragging.

## 21.6.2. Iso-Surface Style

Style settings for all iso-surfaces are handled through the 3D Iso-Surface Details dialog. (These are independent of the style assigned to zones by the Plot Attributes dialog.) The following options are available:

- **Show Iso-Surfaces:** Select this check box to display iso-surfaces.

- **Draw Iso-Surfaces at:** Use this drop-down menu to draw iso-surfaces at each contour level, or up to three specified values.
- **Show Mesh:** Select this check box to display the mesh on iso-surfaces.
- **Mesh Color:** Select a mesh color from the drop-down menu, or choose a custom color or multi-color.
- **Mesh Line Thickness:** Select a line thickness from the drop-down menu, or enter your own number in the text field.
- **Show Contour Flood:** Select this check box to display contour flooding on iso-surfaces.
- **Show Shade:** Select this check box to display shading on iso-surfaces.
- **Shade Color:** Select a shade color from the drop-down menu, or choose a custom color.
- **Use Lighting Effect:** Select this check box to enable the lighting effect drop-down menu where you may choose Paneled or Gouraud shading.
- **Use Surface Translucency:** Select this check box to enable the surface translucency text field, where you may set the surface translucency from one (opaque) to 99 (translucent).

### 21.6.3. Extracting Iso-Surfaces

You may wish to extract existing iso-surfaces to Tecplot zones to retain these surfaces while switching the contour variable to generate a different set of iso-surfaces. Once extracted, the new zones may be plotted like any other zone in which case style is set with the Plot Attributes dialog instead of the 3D Iso-Surface Details dialog.

To extract iso-surfaces to zones, perform the following steps:

1. Add iso-surfaces to your plot as described above.
2. From the Data menu, choose Extract, then choose Iso-Surfaces. The Extract Iso-Surfaces dialog appears, as shown in Figure 21-10.

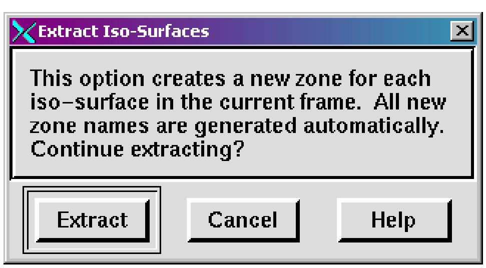



**Figure 21-10.** The Extract Iso-Surfaces dialog.

3. Click Extract to create the new iso-surface zones, one zone for each contour level. All of the variables in the data set are interpolated from the 3-D volume zones to the data points of the iso-surfaces.

Iso-surface zones are FE-surface quadrilateral element-type zones, regardless of the original 3-D volume zone types. The mesh of the iso-surfaces is derived from the mesh of the original zones, so that in regions where the original mesh was coarse, the iso-surface mesh is coarse, and where the original mesh was fine, the iso-surface mesh is fine.

After creating the new iso-surface zones, it is often a good idea to turn off or reconfigure the current settings for iso-surfaces because the new zones will occupy the same physical space as the iso-surfaces.

# 21.7. Slicing Data in 3-D

There are two methods for creating slices:

1. Create slicing planes defined by a constant X-, Y-, or Z-location, or constant I-, J-, or K-index for IJK-ordered zones. These slices are created using either the Slice tool from the sidebar, or the 3D Slice Details option of the Field menu.
2. Extract an arbitrary slice using the Slice from Plane option on the Extract sub-menu of the Data menu. This option allows you to slice through 3-D surface as well as 3-D volume zones.

These operations are separate and each has unique advantages.

## 21.7.1. Defining Slice Planes

Slicing planes defined with the 3D Slice Details dialog or the Slice tool become part of the style of your plot. They do not add to the data set used to create your plot unless you extract them. When you save a layout or stylesheet the information about where the slices are defined will be saved in your file.

Starting and ending slice positions may be defined. Intermediate slice positions between the start and end slice may also be activated. You may generate slices of constant X, Y, or Z, or, if you have IJK-ordered data, you may create slices of constant I, J, or K. Only volume zones may be sliced using this feature. The resulting slices are always 3-D surfaces.

Figure 21-11 shows the pages of the 3D Slice Details dialog. Selecting the Show Slices check box activates the start slice.

**21.7.1.1. The Position Page.** Use the slider to move the start slice, or you may type in the slice position. Activate the end slice and move it with the end slice slider. You may also acti-

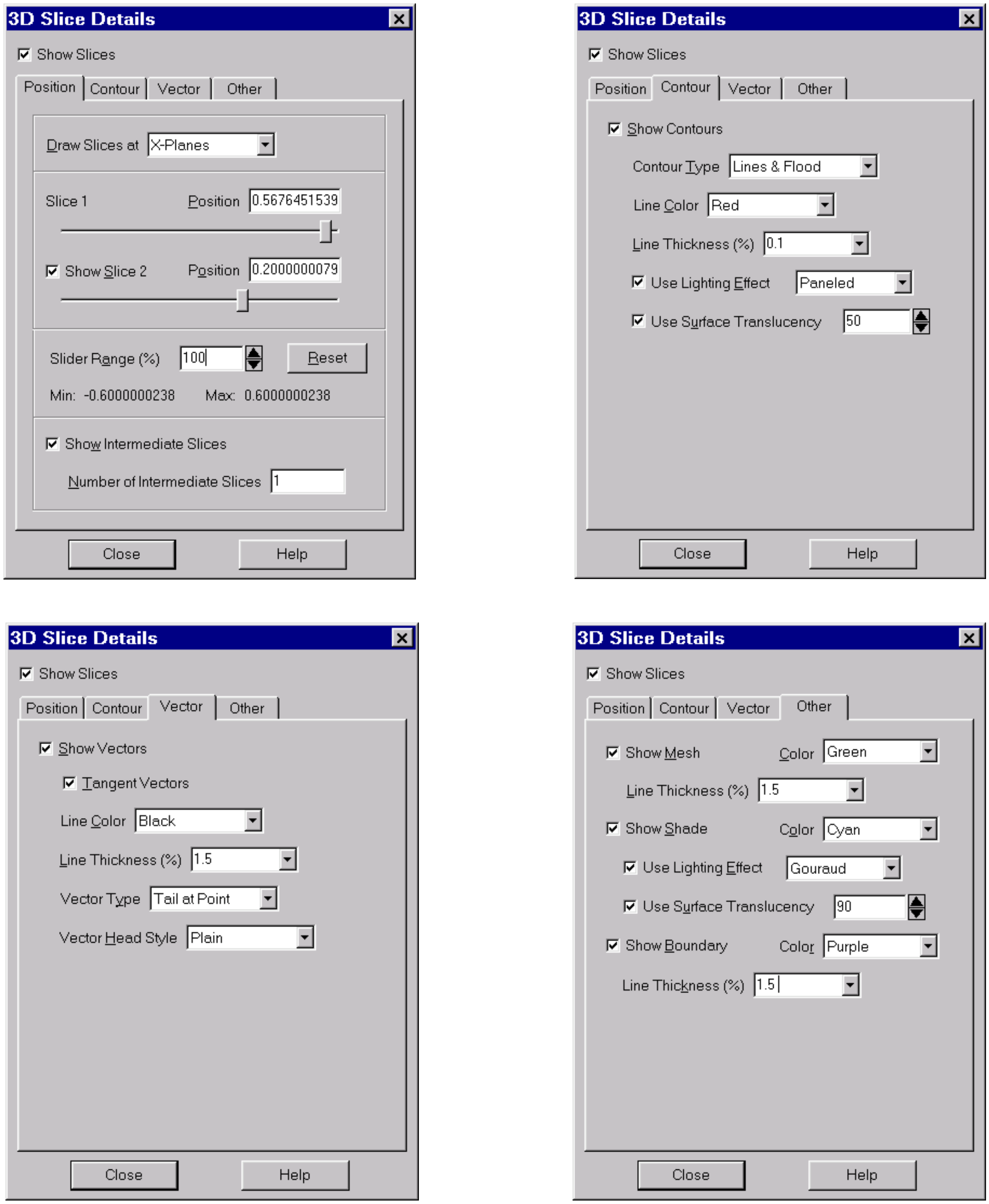



**Figure 21-11.** The pages of the 3D Slice Details dialog, from upper left: the Position page, the Contour page, the Vector page, and the Other page.

vate intermediate slices. Intermediate slices are distributed evenly between the start and end slices.

The following options are available:

- **Show Slices:** Select this check box to enable 3-D slicing.
- **Draw Slices at:** Select which plane to slice on.
- **Slice 1:** Your first slice.
- **Position:** Indicates the current location of your slice. Use the slider to select the position of your slice, or enter a value in the field.
- **Show Slice 2:** Select this check box to show a second slicing plane.
- **Position:** Indicates the current location of your slice. Use the slider to select the position of your slice, or enter a value in the field.
- **Slider Range:** Limit the range for the slides. A value of 100 means the slider range is the same as the range of the axis variable currently being sliced.
- **Show Intermediate Slices:** Select this check box to show intermediate slices between the first and second slices.
- **Number of Intermediate Slices:** Enter the number of intermediate slicing planes in the text field. (Range 1-100.)

**21.7.1.2. The Contour Page.** Use the Contour page to control the contour attributes of your 3-D slices. The following options are available:

- **Show Contours:** Select this check box to show contours.
- **Contour Type:** Select the contour type of the flood from the drop-down menu. Lines, Flood, Lines and Flood are available.
- **Line Color:** Choose the line color from the drop-down menu of Tecplot's basic colors. Multi-Color will color the slice contour lines based on the contour variable.
- **Line Thickness:** Specify the line thickness as a percentage of the frame width. You may enter a value in the text field, or choose one of the values in the drop-down menu.
- **Use Lighting Effect:** Select this check box to enable the lighting effect drop-down menu where you may choose Paneled or Gouraud shading.
- **Use Surface Translucency:** Select this check box to enable the surface translucency text field, where you may set the surface translucency from one (opaque) to 99 (translucent).

**21.7.1.3. The Vector Page.** Use the Vector page to control the vector attributes of your 3-D slices. The following options are available:

- **Show Vectors:** Select this check box to show vectors.
- **Tangent Vectors:** Select to use tangent vectors for your slices.

- **Line Color:** Choose the line color from the drop-down menu of Tecplot's basic colors. Multi-Color will color vectors based on the contour variable. If no contour variable is set when selecting Multi-Color the Contour Variable dialog will appear.
- **Line Thickness:** Specify line thickness as a percentage of the frame width. You may enter a value in the text field, or choose one of the values in the drop-down menu.
- **Vector Type:** Use this drop-down menu to set the vector type for your slices. Choose from Tail at Point, Head at Point, Anchor at Midpoint, and Head Only.
- **Vector Head Style:** Use this drop-down menu to set the vector head style for your slices. Choose from Plain, Filled, and Hollow.

**21.7.1.4. The Other Page.** Use this page to control the mesh, shade, and boundary attributes of your 3-D slices. The following options are available:

- **Show Mesh:** Select this check box to show mesh lines.
- **Color:** Choose the line color from the drop-down menu of Tecplot's basic colors. Multi-Color will color meshes based on the contour variable. If no contour variable is set when selecting Multi-Color the Contour Variable dialog will appear.
- **Line Thickness:** Specify the mesh line thickness as a percentage of the frame width. You may enter a value in the text field, or choose one of the values in the drop-down menu.
- **Show Shade:** Select this check box to show shading on the slice when Show Contour has not been selected or is set to Lines.
- **Color:** Choose the shade color from the drop-down menu of Tecplot's basic colors. Multi-Color will color shading based on the contour variable. If no contour variable is set when selecting Multi-Color the Contour Variable dialog will appear.
- **Use Lighting Effect:** Select this check box to enable the lighting effect drop-down menu where you may choose Paneled or Gouraud shading.
- **Use Surface Translucency:** Select this check box to enable the surface translucency text field, where you may set the surface translucency from one (opaque) to 99 (translucent).
- **Show Boundary:** Select this check box to show selected boundary lines on all slices.
- **Color:** Choose the boundary color from the drop-down menu of Tecplot's basic colors. Multi-Color will color boundaries based on the contour variable. If no contour variable is set when selecting Multi-Color the Contour Variable dialog will appear.
- **Line Thickness:** Specify the boundary thickness as a percentage of the frame width. You may enter a value in the text field, or choose one of the values in the drop-down menu.

**21.7.1.5. Using the Slice Tool.** The Slice tool allows you to position slice planes with your mouse. Select the tool from the sidebar, then click on a surface anywhere in your data. A slice

will be positioned according to the XYZ-location of the nearest surface below where you clicked.

When adding a slice to volume data it is often a good idea to plot the original data using the Shade zone layer and set the translucency to a high level, such as 70 percent. This will allow you to see the outer bounds of your data while placing your slice. It is necessary to see the surface in order to be able to place your slice by mouse-click.

The Slice tool offers mouse and keyboard shortcuts which can greatly speed Tecplot use, especially when working with large amounts of data. These are:

**Click:** Place a start slice.

**Drag:** Move the start slice.

**Shift-click:** Place the end slice

**Shift-drag:** Move the end slice.

**+:** Turn on the start slice if no slices are active, or turns on the end slice if slices are already active.

**- :** Turn off the end slice if the end slice is active, or conversely, turns off the start slice if the end slice is not active.

**I, J, K (ordered zones only):** Switch to slicing constant I-, J-, or K-planes respectively.

**X, Y, Z:** Switch to slicing constant X-, Y, or Z-planes respectively.

**1-9:** Activate intermediate slices and set the number of intermediate slices to the number entered.

**0:** Turn off intermediate slices.

**Alt-click/Alt-drag:** Determine the XYZ-location by ignoring zones and looking only at derived volume objects (streamtraces, slices, iso-surfaces, slices).

## 21.7.2. Extracting Slices

In most cases it is not necessary to extract slices to zones. Most existing slice features allow you to set almost any style. There are cases where you may need to display multiple sets of slices in various directions, so it is necessary to extract at least some of the slices to zones.

**21.7.2.1. Extracting Pre-Defined Slices.** To extract slices that you have pre-defined with the Slice tool or the 3D Slice Details dialog choose the Current 3D Slices option from the Extract sub-menu of the Data menu. This option will create a separate zone for each slice plane.

**21.7.2.2. Extracting an Arbitrary Slice.** To extract a slice at an arbitrary orientation, or to slice a 3-D surface instead of a volume, use the Slice from Plane option from the Extract sub-menu of the Data menu.

Specify any of four different types of cutting planes, as follows:

- **Arbitrary:** An arbitrary cutting plane. You may specify the position and orientation of the cutting plane using three points or an origin and a normal vector, or you can interactively place and rotate the cutting plane using the controls in the Extract Slice dialog.
- **Constant X:** A cutting plane of constant X-value. You may specify the X-value either by entering a value, or using a position slider.
- **Constant Y:** As Constant X above, but for a cutting plane of constant Y-value.
- **Constant Z:** As Constant X above, but for a cutting plane of constant Z-value.

To slice a 3-D zone with a plane:

1. From the Data menu, choose Extract, then choose the Slice from Plane option. The Extract Slice from Plane dialog appears as in Figure 21-12.

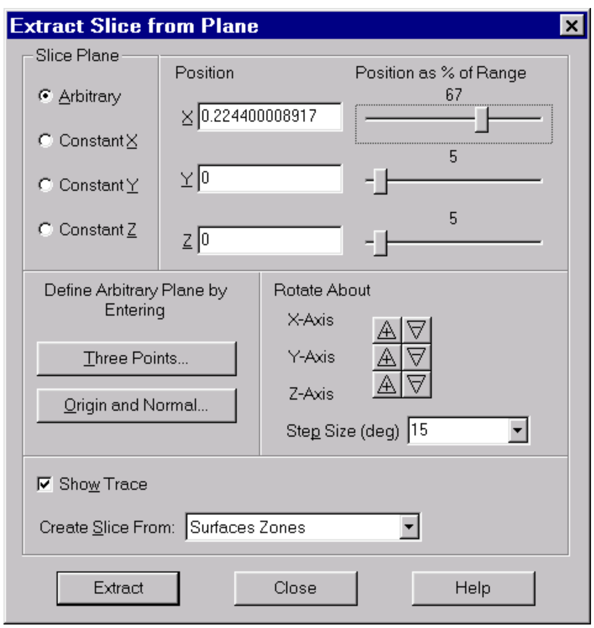



**Figure 21-12.** The Extract Slice from Plane dialog.

2. Choose the option button corresponding to your desired slice plane (Arbitrary, Constant X, Constant Y, or Constant Z).
3. If you choose Arbitrary as your cutting plane, you can either use the Position sliders and Rotate About buttons to position the cutting plane, or choose one of the following buttons:
   - **Three Points:** Calls up the Enter Three Points dialog, in which you specify the cutting plane by entering the X-, Y-, and Z-coordinates of three points on the cutting plane (nine numbers in all). These points must form a triangle; they cannot be coincident or collinear.
   - **Origin and Normal:** Calls up the Enter Slice Origin and Normal dialog in which you specify the cutting plane by entering the coordinates of a point and the components of a normal vector. Using this option, you enter six numbers to specify the cutting plane: the X-, Y-, and Z-coordinates of a point on the cutting plane (called the slice origin), and the X-, Y-, and Z-components of a vector normal to the cutting plane (called the slice normal).

   Use the X,Y, and Z Position sliders (or the associated text fields) to move the cutting plane's slice origin. A representation of the slice plane is shown in the workspace. Use Rotate About to rotate the slice plane about the slice origin.
4. To see a "trace" of the current slice, select the Show Trace check box. If Show Trace is selected, Tecplot draws an approximation of the intersection of the slicing plane with the active 3-D zones. For finite-element zones, the trace in fact draws all line segments of the intersections of the slicing plane with the cells in the zone. For IJK-ordered data, the trace is simply the line resulting from the intersection of the slicing plane and the outer surface of the zone. If Show Trace is not selected, Tecplot simply draws the intersection of the slicing plane with the axis box.
5. Choose to create slices from volume zones, surface zones, or surfaces of a volume zone. A slice from a volume zone will create a plane. A slice from a surface zone, or the surface of a volume zone, will be as a line or curve.
6. Click Extract to extract the slice as a finite-element surface zone.

Once you have created the slice zone, you may plot it, write it out to a data file, delete it, etc. It is the same as any zone that was read into Tecplot. If you slice volume zones the resulting slice zones created are finite-element surface, quadrilateral element-types. If you slice surface zones the resulting zones are finite-element surface, triangle element types.

See Figure 21-13 for an example of a zone created by a slice.

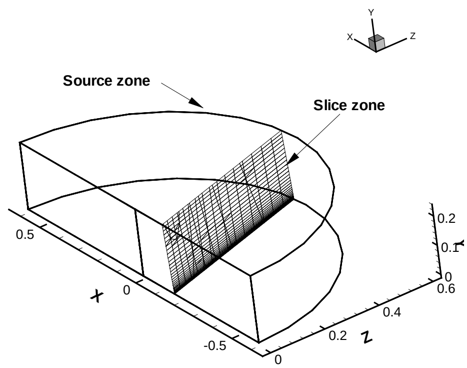



**Figure 21-13.** Zone extracted by slicing 3-D volume zone.

# 21.8. Creating Special 3-D Volume Plots

Special 3-D volume plots include fence plots, so called because they look like a flat plane divided by fences, and analytic iso-surface plots, which plot iso-surfaces for analytic functions such as $F(x,y,z)=2x^2-3y^2-7z^2$, and cutaway plots, which are discussed in Chapter 27, “Blanking.”

## 21.8.1. Fence Plots

A fence plot is a plot of planes of a 3-D data field. These planes may be IJ-ordered zones, or combinations of I-, J-, and K-planes of an IJK-ordered zone. In particular, the “bottom” plane of the plot is plotted, plus a few planes that are perpendicular to this plane. These perpendicular planes are the “fences.” Typically, flooded contours are plotted on each plane. An example fence plot is shown in Figure 21-14.

Creating a fence plot with IJK-ordered data is simple; just perform the following steps:

1. Read in the IJK-ordered data set.
2. Select the Contour zone layer check box on the sidebar. (This will initially be an iso-surface plot.)
3. Deselect the Mesh zone layer.
4. From the sidebar click Plot Attributes. The Plot Attributes dialog appears.
5. From the Volume page set Surfaces to Plot to I&J&K Planes.
6. For each of I, J, and K, set the index range as appropriate. In Figure 21-14, the ranges chosen were as follows:

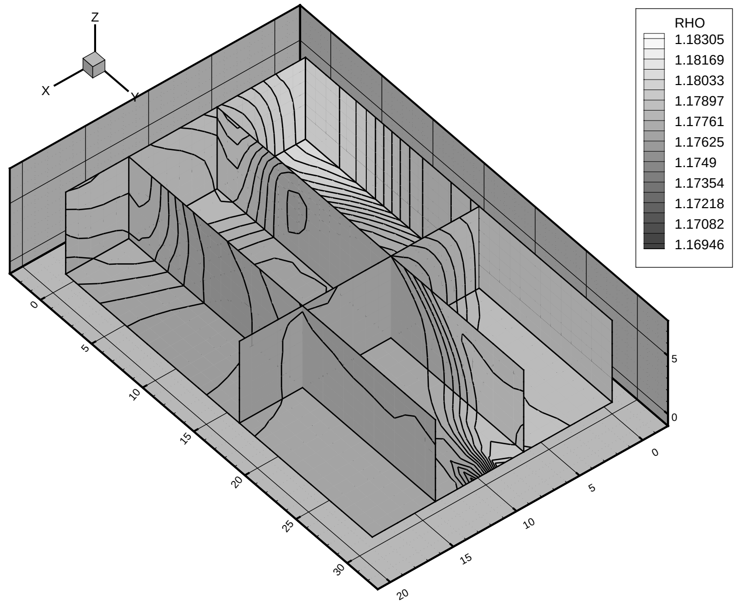



**Figure 21-14.** A fence plot.

- **I-planes:** *Start=1*, *End=1*, *Skip=1* (the bottom plane)
- **J-planes:** *Start=1*, *End=Mx*, *Skip=7* (the three parallel planes)
- **K-planes:** *Start=1*, *End=Mx*, *Skip=17* (the two parallel planes)

7. From the Contour Attributes dialog, set the contour plot type to Flood or Both Lines and Flood. Set the Flood Translucency to Medium.
8. Select the Shade zone layer.
9. Redraw your plot. You should see a fence plot similar to that in Figure 21-14.

You can also create fence plots using IJ-ordered zones. For the best effect, the plotted zones should be perpendicular to each other when plotted in 3-D. For example, you can create a fence plot from the planes extracted using the procedure in Section 21.5, “Extracting I-, J-, and K-Planes.”

### 21.8.2. Analytic Iso-Surface Plots

Using Tecplot's data manipulation tools, you can create iso-surface plots of 3-D volume analytic functions such as $F(x,y,z)=2x^2-3y^2-7z^2$. An iso-surface plot of this function in the range $x=0$ to $x=1$, $y=0$ to $y=1$, and $z=0$ to $z=1$ is shown in Figure 21-15.

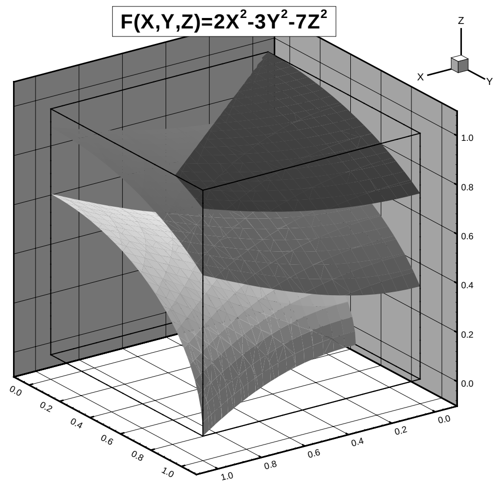



**Figure 21-15.** Iso-surface plot of an analytic function.

To create an iso-surface plot of an analytic function, perform the following steps:

1. From the File menu, choose New Layout to clear the workspace.
2. From the Data menu, choose Create Zone, and then choose Rectangular. The Create Rectangular Zone dialog appears.
3. Enter the dimensions of I, J, and K. These will be, respectively, the number of points plotted in the X-, Y-, and Z-directions. Figure 21-15 was plotted using a dimension of 20 for each index.
4. Enter zero for each of XMin, YMin, and ZMin; enter one for each of XMax, YMax, and ZMax.
5. From the Data menu, choose Alter, then choose Specify Equations. The Specify Equations dialog appears.
6. In the Equation(s) text field, enter the following equation:

```
V4 = 2*X*X - 3*Y*Y - 7*Z*Z
```

7. Click Compute to create the new variable.
8. Deselect the Mesh zone layer check box on the sidebar.
9. Bring up the 3D Iso-Surface Details dialog from the Field menu. When the Select Variable dialog appears, accept the default, v4, as the contour variable.
10. Turn on Iso-Surfaces.

# CHAPTER 22 Printing Plots

Printing your plot is the process of sending the plot image to an output device, print spooler, or a file. Output devices include printers, plotters, film recorders, and typesetting machines. You can print any plot to a file, instead of having it print directly on the printer or plotter. If you are creating files for use in another program, you should use Tecplot's Export menu to create your files—the Export menu includes all the supported print file types, as well as several standard graphics formats such as TIFF, WMF, and EPS. See Chapter 23, "Exporting Plots," for complete details.

## 22.1. Printing a Plot

To print a plot, select Print from the File menu. From the Print dialog you can specify whether the output is sent directly to the printer or print spooler or to a file, and also specify the number of copies (available for Motif only). When you click OK, everything visible on the Tecplot paper is printed, either on the printer or to a file. The Print dialog is shown in Figure 22-1.

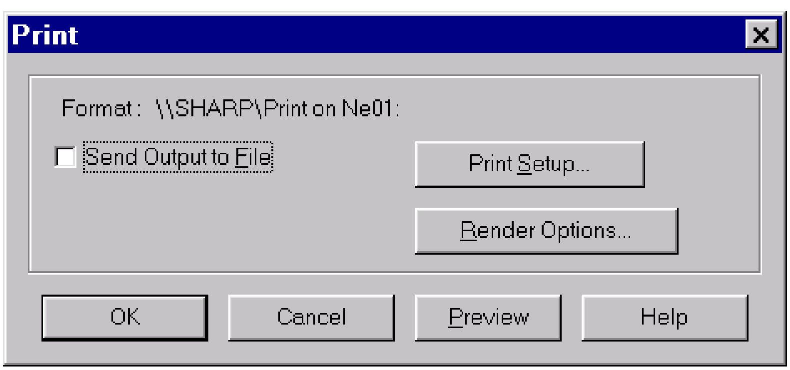



**Figure 22-1.** The Print dialog under Windows.

If you select the Send Output to File check box, the Print to a File dialog appears and you may select the name of the file to create.

The Print dialog lets you specify options for setting up your printer by accessing the Print Setup dialog as discussed in Section 22.3, “Setting Up Your Printer,” and for setting additional controls for preparing the plot being sent to the printer via the Print Render Options dialog as discussed in Section 22.4, “Print Render Options.” The Print dialog also provides access to Print Preview, which is discussed in Section 22.5, “Print Preview.”

At any time in your Tecplot session, you can set various parameters relating to the paper, including paper size and orientation, using the Paper Setup dialog and the Print Setup dialog. A change to your paper settings in either the Paper Setup dialog or the Print Setup dialog will automatically update the other.

## 22.2. Setting Up Your Paper

You may use the Print Setup dialog under Windows, or the Paper Setup dialog, to specify your paper size and orientation. The Print Setup dialog is called up by clicking Print Setup on the Print dialog. The Print dialog is accessed from the File menu’s Print option. The Paper Setup dialog is accessed from the File menu’s Paper Setup option.

A change to your paper settings in either the Paper Setup dialog or the Print Setup dialog will automatically update the other.

### 22.2.1. Using the Print Setup Dialog under Windows

The Print Setup dialog is preferable under Windows for setting up your paper. It lists all the paper sizes your printer supports. Tecplot can produce output to fit virtually any paper size. You may select a paper size using the Print Setup dialog’s Size drop-down menu.

Print Setup allows you to specify the paper source tray if your printer has multiple paper trays. Do this by choosing a tray from the Print Setup dialog’s Source drop-down menu.

You can choose either Portrait or Landscape paper orientation from the Print Setup dialog. In Portrait orientation, the long axis of the paper is aligned with the vertical axis of the plot. In Landscape orientation, the long axis of the paper is aligned with the horizontal axis of the plot.

The Print Setup dialog is shown in Figure 22-2.

### 22.2.2. Using the Paper Setup Dialog

To adjust the paper size, orientation, and background color for your plots, select the Paper Setup option from the File menu. The current settings for these options are reflected in the rep-

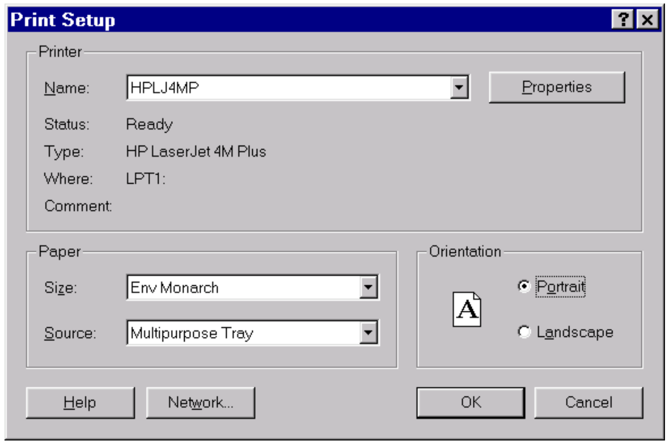



**Figure 22-2.** The Print Setup dialog under Windows.

resentation of the paper in the workspace. (To view the paper, select the Show Paper on Screen check box in either the Paper Setup dialog or the Ruler/Grid dialog under the Workspace menu. This check box is selected by default.)

The Paper Setup dialog, in contrast with the Print Setup dialog under Windows, offers you only six paper sizes. These may not be compatible with the paper sizes your printer supports. You cannot select from multiple paper trays with the Paper Setup dialog. You may set screen display options and fill colors with the Paper Setup dialog. The Paper Setup dialog is shown in Figure 22-3.

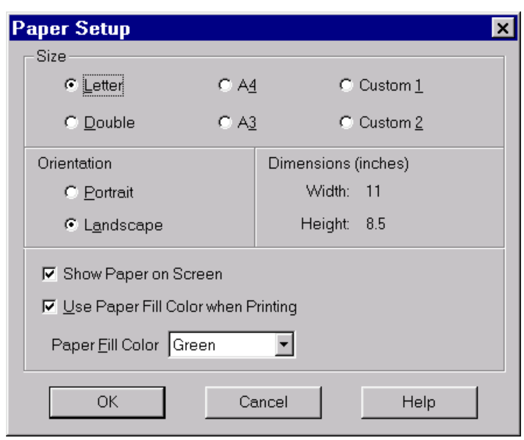



**Figure 22-3.** The Paper Setup dialog.

The following options are available in the Paper Setup dialog:

- **Size:** Choose the size of the paper from the following six selections:
  - Letter (8.5 x 11 inches).
  - Double (11 x 17 inches).
  - A4 (21x 29.7 cm).
  - A3 (29.7 x 42 cm).
  - Custom 1 (8.5 x 14 inches).
  - Custom 2 (8 x 10 inches).

  Under Windows, paper size Custom 2 is overwritten with the size selected in Print Setup if that size does not exist in Tecplot.

  You can customize all six paper sizes in the configuration file, as well as their hard-clip limits. The hard-clip limits are the lines on the edges of the paper that show where your printer cannot print. You can set the hard-clip limits to larger values for use as guides in placing your plots on the paper.
- **Orientation:** Choose the paper orientation. You have two options: Portrait and Landscape. In Portrait orientation, the long axis of the paper is aligned with the vertical axis of the plot. In Landscape orientation, the long axis of the paper is aligned with the horizontal axis of the plot.
- **Paper Fill Color:** Select a color to use for the paper background. This color is used to display the paper in the workspace. You can select the check box Use Paper Fill Color when Printing to have Tecplot print this background color on the hard-copy as well.

# 22.3. Setting Up Your Printer

You use the Print Setup dialog to set up Tecplot for printing on a particular printer. The available options are different on Windows and Motif systems. The Print Setup dialog is called up by clicking Print Setup on the Print dialog. The Print dialog is accessed from the File menu's Print option.

## 22.3.1. Setting Up Windows Printing

To set up for printing on Windows systems, select the Print option from the File menu. The Print dialog will appear, which was shown in Figure 22-1. Click Print Setup to launch the Print Setup dialog, shown in Figure 22-2.

You may choose to use the printer and specifications presented, or you may click Print Setup or (Print) Render Options to customize your printing.

You may choose to use the Windows default printer, or choose from any currently installed printers. To change to another installed printer, click on the Name drop-down menu and select another printer from the list.

### 22.3.2. Setting Up Motif Printing

Setting up to print under Motif includes the following tasks:

- Specifying a spool command, if you are using a print spooler. This may include specifying a device-dependent startup string to condition the output device for the Tecplot output, or a mopup string to reset the output device upon completion of plotting.
- Specifying the precision of the output for those formats which support variable precision.
- Assigning pen colors to pens for pen plotters, if applicable.

You perform most of these tasks from the Motif version of the Print Setup dialog, accessed from the Print option of the File menu, and shown in Figure 22-4. Some of the Print Setup dialog options launch additional dialogs, which are discussed at the appropriate places in the following sections.

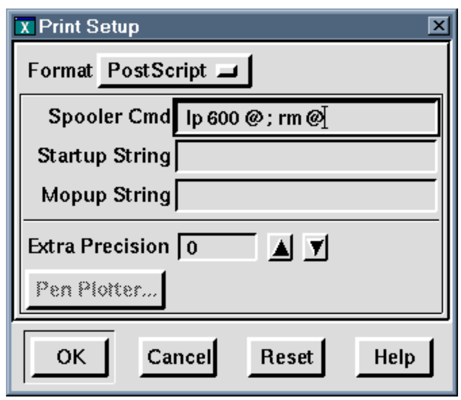



**Figure 22-4.** The Motif version of the Print Setup dialog.

**22.3.2.1. Choosing a Print Format.** In Tecplot, you can choose from any of the following print formats:

- **PostScript (color or monochrome):** PostScript is the recommended output format, since it supports all Tecplot fonts (including Greek and Math), color flooding (or gray-scale flooding), hidden surface (or line) removal, and overlaid frames (plots).

- **HP-GL (line color only):** The HP-GL format generates output for most HP pen plotters as well as other pen plotters that emulate the HP-GL language. Screen colors are mapped to pen numbers on the plotter. HP-GL output can be imported into some programs, such as WordPerfect, but it has some limitations. It does not support contour flooding, multi-coloring, or hidden-surface removal, and it is restricted to using the Tecplot stroke (screen) fonts.
- **HP-GL/2 (color or monochrome):** The HP-GL/2 format can be used for HP-GL pen plotters as well as the HP LaserJet III, LaserJet 4, and PaintJet XL printers. When used with a supported printer (as opposed to a pen plotter), HP-GL/2 can show contour flooding, multi-coloring and hidden-surface removal, but plots are always limited to using the Tecplot stroke fonts.

To choose the print format:

1. Call up the Print Setup dialog by clicking Print Setup on the Print dialog, accessed from the File menu.
2. Choose the desired format from the Format drop-down menu.

If your printer is incompatible with all of the above formats, you might consider a PostScript converter such as the freeware program Ghostscript. These programs interpret PostScript output and translate it to the native languages for dozens of supported devices.

**22.3.2.2. Specifying a Spool Command.** Printers on most UNIX systems are accessed via print spoolers that manage the print queue. Under UNIX, you typically use either the **`lp`** or **`lpr`** commands to send files to the print spooler. There may be command-line options that need to be set on your system, as well, such as a flag to specify a particular printer. You use the Print Setup dialog to specify the appropriate spool command for your system.

To specify the spool command for your system:

1. Call up the Print Setup dialog, and choose the desired format. (Spool commands will most likely be different for different print formats, and Tecplot stores one spool command for each print format.)
2. In the Spooler Cmd text field, enter the appropriate spool command for your system, using the `@` symbol to represent a file name.

For example, suppose you routinely use the following spool command to print a file named **`myfile.ps`**: “**`lpr -m -r myfile.ps`**.” The appropriate spooler command to enter in the Spooler Cmd field is then “**`lpr -m -r @`**.”

When printing to a spooler, Tecplot creates temporary files with names of the form **`tp??????`**, where the **`?`**s are randomly generated characters. Tecplot does not delete these temporary files automatically; commands to do so should be included in your spool command. In our example, the **`-r`** flag says to remove the file when done.

**22.3.2.3. Specifying Startup and Mopup Strings.** A startup string is an initialization string that sets up your output device to accept the plot created by Tecplot. A mopup string is a reset signal that tells your output device that the special output has ended. For most devices no startup or mopup strings are needed. However, some common devices, such as the HP LaserJet III when printing HP-GL/2, require both startup and mopup strings.

To specify a startup or mopup string:

1. Call up the Print Setup dialog, and choose the desired format. (Startup and mopup strings will be different for each format, and Tecplot stores one startup and mopup string for each print format.)
2. Enter the appropriate startup string or mopup string in the appropriate text field. Special characters are generated by using Macro Codes (such as “**%E**” for the escape character and “**^***nnn*” for any ASCII character with a decimal ordinal value of *nnn*). Check your printer documentation for the appropriate strings. For example, with some HP-GL implementations, the HP-GL startup string must be set to the following:
   **$E.J$E.N;19:$E.I81;;17:**

   The HP LaserJet III requires the HP-GL/2 startup and mopup strings shown below:
   - Startup String: **$E%-1B**
   - Mopup String: **$E%0A^012**

**22.3.2.4. Controlling Printing Precision.** For PostScript and HP-GL/2 output, you can control the numerical precision used in your print files. Print files contain numbers that define sizes and positions of pieces of the plot on the output paper. These numbers are defined as integers between zero and about 8,000. Usually, this provides sufficient resolution for most output devices. Occasionally, you may need more resolution. For example, printing to a high-resolution output device like a Linotronic typesetter may require more precision; making print output with very small cells or elements may also require more precision.

To increase the precision of the output, increase the value in the Extra Precision field of the Print Setup dialog. You specify one Extra Precision value for all formats that supports precision control. The precision is defined as the number of digits to the right of the decimal. Normally, precision is zero. The disadvantage of setting precision high is that the print files increase in size. The higher the Extra Precision setting, the larger your print files, but the more accurate the plot. Numbers above two are not normally required unless you need extremely fine resolution. The maximum setting for the precision is eight.

**22.3.2.5. Configuring Pen Plotters.** If you are using a pen plotter, you can use the Pen Plotter Device Configuration dialog shown in Figure 22-5 to specify plotter speed and pen assignments for particular colors and Tecplot object types. You access this dialog by clicking Pen Plotter on the HP-GL page of the Print Setup dialog. You can associate each of Tecplot’s basic colors with any of the plotter’s pens.

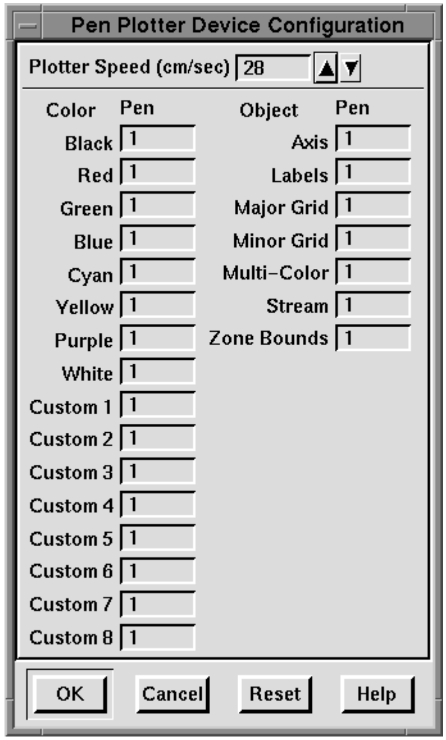



**Figure 22-5.** The Pen Plotter Device Configuration dialog. (Motif only.)

You may also specify that the following object types be associated with a particular pen:

- Axes.
- Tick mark labels.
- Major grid lines.
- Minor grid lines.
- Objects colored with the multi-color option.
- Streamtraces.
- Zone boundaries.

To specify a pen assignment, simply enter the number of the desired pen in the appropriate color or object field.

To specify a plotter speed, either enter a value in the Plotter Speed field, or use the up and down arrow buttons to increment and decrement the current value.

Click OK to accept your changes, Cancel to quit with no changes, or Reset to reset the configuration to its last saved value.

The pen plotter configuration settings only affect the output only when the print format is HP-GL; they have no effect on PostScript or HP-GL/2. There are no settings for Flood Pens because flooding is not supported under HP-GL. The default is for all lines and text to use Pen 1.

## 22.4. Print Render Options

Clicking the Print dialog's (Print) Render Options calls up the Print Render Options dialog, shown in Figure 22-6.

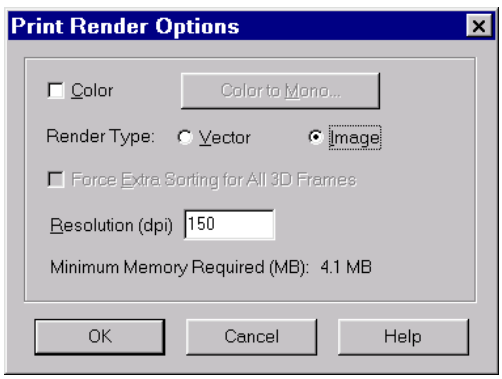



**Figure 22-6.** The Print Render Options dialog under Windows.

The Print Render Options dialog offers you the following choices:

- **Color:** Select this check box for color output; deselect the check box for monochrome output.
- **Color to Mono:** This option is available when the Vector option is selected, but the Color check box is not selected. Clicking Color to Mono, when it is available, calls up the Monochrome/Gray Scale Mappings dialog, which you use to specify how colors map to gray scales when creating monochrome (black and white) output from color plots. For more information see Section 22.4.1, "Specifying Color Mappings for Monochrome Printing."

- **Vector:** Select this option to create print output using the drawing commands of the printer. The printer renders the plot, yielding higher resolution, but some plot options, such as translucency, are not available.
- **Image:** Select this option to create print output using an image. Rendering is done by Tecplot at the specified resolution, usually less than the printer's resolution. However, all plot options are available.
- **Force Extra Sorting for all 3D Frames:** This option is available when the Vector option has been selected. Selecting this check box will cause Tecplot to use extra sorting in all 3-D frames. This overrides the setting in the Advanced 3D dialog. If this check box is not selected, Tecplot will choose sorting algorithms based on the Advanced 3D dialog options that were chosen for each frame. When printing 3-D plots in a vector graphics format, Tecplot must sort the objects so that it can draw those farthest from the screen first and those closest to the screen last. By default, Tecplot uses a quick sorting algorithm. This is not always accurate and does not detect problems, such as intersecting objects. If Extra Sorting is selected, Tecplot uses a slower, more accurate approach that detects problems.
- **Resolution (dpi):** Available when the Image option is selected. Enter the resolution in terms of dpi in the text field. Larger resolutions may result in an out-of-memory condition, or produce very large files. Smaller resolutions may yield less-attractive output images.

The Print Render Options dialog also indicates the amount of memory your final output will require when the selected Render Type is Image.

## 22.4.1. Specifying Color Mappings for Monochrome Printing

When you create monochrome plots from Tecplot, particularly if you have been working in color on your screen, you can specify how Tecplot maps various colors to shades of black, white, and gray. With the standard configuration, all colors of lines and text (including white) are mapped to black, and the colors of flooded regions (geometries, frame background, grid background color) are mapped to various gray scale values. Using the Monochrome/Gray Scale Mappings dialog, you specify how lines, text, and flooded objects on the screen are mapped to shades of gray in the print output. Shades of gray are specified as a percentage (zero to 100) of white. These mappings affect only monochrome formatted output. Color output formats use screen colors.

To specify the color to gray scale mappings:

1. Call up the Print Setup dialog and choose the desired format. Deselect the Color check box if it exists.
2. Click Color to Mono to bring up the Monochrome/Gray Scale Mappings dialog shown in Figure 22-7. This is a global dialog—the settings apply to all monochrome output formats.

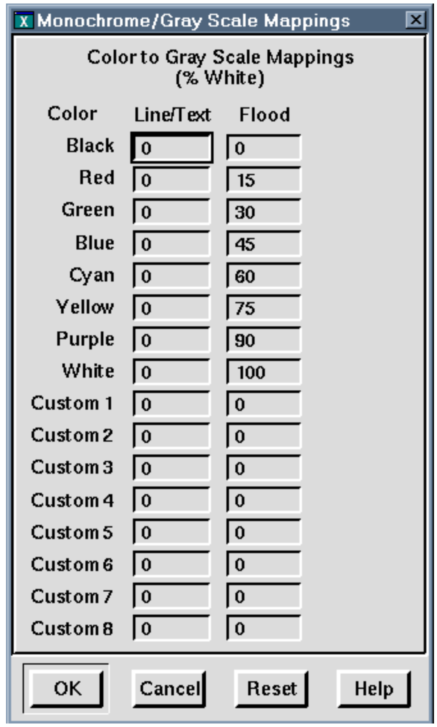



**Figure 22-7.** The Monochrome/Gray Scale Mappings dialog.

3. In the Line/Text column, assign a percentage of white for each of Tecplot's basic colors. This percentage represents the shade of gray in which lines and text of that color will be printed (zero represents black, 100 represents white).
4. In the Flood column, assign a percentage of white for each of Tecplot's basic colors. This percentage represents the shade of gray in which color filling of that color will be printed (zero represents black, 100 represents white).
5. Click OK to accept your changes, Reset to restore the most recently saved values, or Cancel to quit with no changes.

The colors used for contour flooding and multi-coloring are converted to gray scale values according to the NTSC standard:

*GRAY SCALE= 0.3*RED + 0.59*GREEN + 0.11*BLUE*

For the best black and white plots, Amtec recommends that you switch the Tecplot color map to Gray Scale for printing on monochrome printers. See Section 11.5, “Controlling the Global Color Map,” for details.

Specifying how colors map to gray scales when creating monochrome (black and white) output from color plots.

## 22.5. Print Preview

A preview of your screen image as it will be rendered for the printer may be generated by selecting Print Preview from the File menu. The preview image may be accessed by clicking Preview on the Print dialog, which is shown in Figure 22-1. There are several reasons for viewing the print preview image prior to sending the plot to the printer. They are associated with the image quality and reduced image content that can be supported for vector graphics printer formats such as PostScript.

As discussed in Section 22.4, “Print Render Options,” the default sorting algorithm used by Tecplot may have problems with intersecting objects. This will typically not show up in the OpenGL-rendered screen image. However, sorting errors may occur for vector print output. These will be visible in the preview. The Print Preview option provides access to the Print Render Options dialog, where you may improve sorting by selecting Force Extra Sorting for All 3D Frames. If extra sorting does not solve the problem, the only option available is to export the plot using an image format, discussed in Chapter 23, “Exporting Plots.” By increasing the resolution for an image format you can obtain a quality comparable to PostScript without the sorting errors.

Vector graphics formats do not support translucency, contour flooding with Gouraud shading, or contour flooding using the continuous color distribution method (which is only available with OpenGL). Print Preview will not display translucency. Gouraud shading for contour flooding will be reduced to Paneled shading. Continuous color flooding will be reduced to color flooding with average-cell color. When you print, warning messages will be displayed to advise you about the unsupported plot styles. The resulting printed output will closely match the print preview image.

# CHAPTER 23 Exporting Plots

In Chapter 22, "Printing Plots," we stated that all Tecplot plots could be printed to files, as well as directly to printers. Print files are useful only for printing at a later time. Sometimes, however, you want to create files for use in other applications, such as plots to be included in a word-processor document, or to be edited by a graphics program. Sometimes, you can use print files for these purposes. More often, however, you need plots in different formats. Use the Export dialog under the File menu to create files for export into other applications.

Tecplot supports two main types of export files—vector graphics and raster graphics. Vector graphics have device-independent resolution, but they have the same limitations as vector print output.

With vector graphics, the export file specifies a series of lines on the paper by specifying start and end points. The importing application then simply draws a line from the start point to the end point. The Tecplot Export menu supports the following vector plot formats:

- **PostScript (PS):** Suitable for direct printing, but unsuitable for export to other applications. It is recommended that you use the Encapsulated PostScript (EPS) format for importing into other applications.
- **Encapsulated PostScript (EPS):** Special type of PostScript file designed for inclusion in other applications.
- **Windows Metafiles (WMF):** Used to import into various Windows applications.
- **HP-GL:** Mainly for pen plotters. Screen colors are mapped to pen numbers. You can import these files into some applications.
- **HP-GL/2:** Mainly for pen plotters, HP LaserJet II or IV, PaintJet XL.

Tecplot also allows you to create a raster or bit-mapped image. In Tecplot's nomenclature there are called image formats. Raster files contain images of plots defined as an array of pixels. Each pixel is a dot and may have one or more bits of information that define the color or gray-scale value of the dot. Tecplot creates its raster, or bitmap, files by scanning the image, reading the color of each pixel, and writing it to a file. You define a region of the workspace, and

Tecplot creates a file of the specified region in the image format that you specify, scaled to the requested size of the image. Tecplot supports the following image formats:

- **PNG:** Portable Network Graphic.
- **BMP:** Windows Bitmap.
- **AVI**: Audio Visual Interleaved, a Windows movie file format. You may set the number of frames per second.
- **TIFF:** Tagged Image File Format. You may export a color or gray scale image. For gray scale, you may select an image depth (see below for details).
- **X-Windows:** An image in “xwd” (X-Window Raster) format.
- **Sun Raster:** A Sun Microsystems’ Sun Raster image.
- **Raster Metafile:** A NASA Raster Metafile. Used for creating movies for Framer.
- **PostScript Image:** A PostScript screen image. This is a raster EPS file suitable for printout or import into other applications. You may choose to export a color or monochrome image. In most cases, you want to use the vector format of PS or EPS instead of creating a PostScript image.

Of all the image formats only **`.rm`** and **`.avi`** allow you to append multiple images to a file.

**Note:** On Windows, you cannot create color bitmaps from Tecplot unless your graphics card supports the display of at least 256 colors. In addition to the Export dialog, the Windows version allows you to export a vector-based image (**`.wmf`**) or a raster-based image (**`.bmp`**) directly to the Clipboard. If you are running Windows with only 256 colors, exporting may be very slow.

## 23.1. Creating a File for Export

The basic procedure for creating an export file is the same whether you are creating a vector format or image format file:

1. From the File menu, select Export. This calls up the Select Export Format dialog, shown in Figure 23-1.
2. Select a format from the Format list.
3. Set format-specific options. The vector formats draw from one set of options and the image formats from another, so we will consider the two types separately in discussing these options.
4. When you click OK in the Select Export Format dialog, you must specify a file name for the export file.

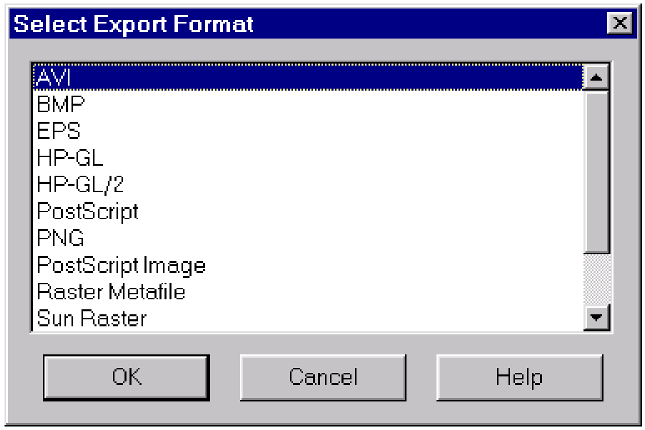



**Figure 23-1.** The Select Export Format dialog.

## 23.2. Creating Vector Export Files

For any of the three native Tecplot print formats (HP-GL, HP-GL/2, PostScript), the process for creating an export file is similar to printing to a file on a Motif system. (Windows users should review the discussion of printing under Motif in Section 22.3.2, “Setting Up Motif Printing.” ) Print formats are supplied as options on the Export menu for convenience.

Available on all vector export formats is Tecplot’s Force Extra Sorting for All 3D Frames option. Selecting this check box will cause Tecplot to use extra sorting in all 3-D frames. This overrides the setting in the Advanced 3D dialog. If this check box is not selected, Tecplot will choose sorting algorithms based on the Advanced 3D dialog options for a given frame. When exporting 3-D plots in a vector graphics format, Tecplot must sort the objects so that it can draw those farthest from the screen first and those closest to the screen last. By default, Tecplot uses a quick sorting algorithm. This is not always accurate and does not detect problems, such as intersecting objects. If Extra Sorting is selected, Tecplot uses a slower, more accurate approach that detects problems.

### 23.2.1. Encapsulated PostScript (EPS)

Encapsulated PostScript (EPS) format can be imported into many programs (desktop publishing and word processing for instance). EPS format files may include an embedded bit-map image that displays on the screen in many of the programs that import EPS files. There are several different forms of bit images from which you can choose. Tecplot supports TIFF (for most PC programs), PostScript Version 2, and a special EPS format for Adobe’s FrameMaker

program. The embedded bit image only approximates the actual PostScript image. The default preview image type is TIFF. The EPS Exporter dialog is shown in Figure 23-2.

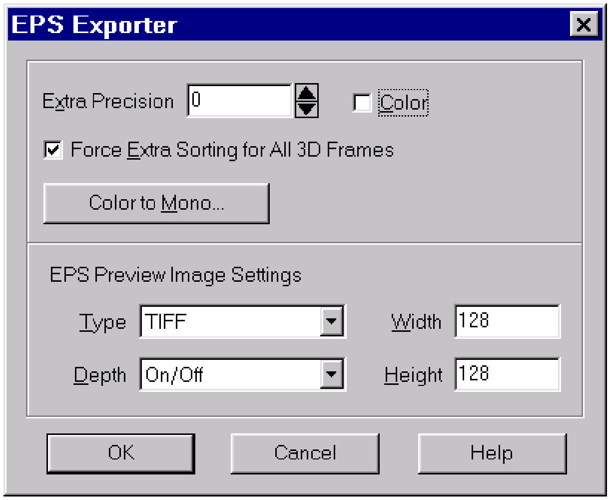



**Figure 23-2.** The EPS Exporter dialog.

The following options are available:

- **Extra Precision:** Specify the number of decimal places to which size and position parameters are carried. The number of decimal places added to the integer value will be used as coordinates for the resulting vector-based print output. Use the increase or decrease arrows, or enter a value in the text field.
- **Color:** Allows you to create color Encapsulated PostScript images.
- **Force Extra Sorting for All 3D Frames:** Selecting this check box will cause Tecplot to use extra sorting in all 3-D frames.
- **Color to Mono:** This calls up the Monochrome/Gray Scale Mappings dialog. You may examine or modify the mappings from colors used for screen lines or text and floods to gray scales used in monochrome plotting. By default, all lines and text are converted to black. Flood colors are mapped to a gray scale except for custom colors, which are set to black. This option is only available when exporting non-color (gray scale) images.

In addition, you may choose the type of preview image included in your EPS files (both color and monochrome). A preview image is a rough sketch of your print file used by importing programs. When you choose an EPS image, you may also specify its resolution (in pixels) in the

width and height fields. Low values make the preview image poor. High values can make the EPS file large.

Options for preview images include:

- **None:** No embedded image information is added to the EPS file. This is good for importing into applications that do not use image information.
- **TIFF:** Include a monochrome/gray-scale TIFF image in the EPS file. (Color preview images are not available.) This is the most common image format. You may specify a image depth for the TIFF image, as described in Section 23.3.4, “Creating TIFF Images.”
- **EPSIV2:** Include a monochrome (one bit per pixel) Encapsulated PostScript Version 2 image. This is also a common image type of EPS Print Files.

## 23.2.2. Windows Metafile (WMF) Export

Windows and UNIX versions of Tecplot can export a Tecplot image in Windows Metafiles (WMF) format. These “placeable metafiles” can be imported into many other applications. Selecting Windows Metafile from the Select Export Format list brings up the Windows Metafile Exporter dialog, shown in Figure 23-3.

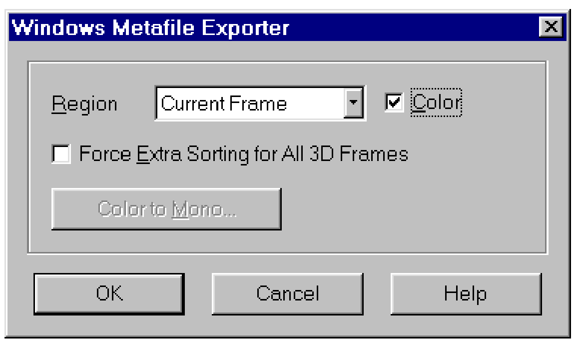



**Figure 23-3.** The Windows Metafile Exporter dialog.

The following options are available:

- **Region:** By default, the image region is the current frame, but you can also choose the smallest rectangle containing all frames, or the full workspace. To specify the region, choose the desired option (Current Frame, All Frames, or Work Area) from the Region drop-down.
- **Color:** Allows you to create color Windows Metafile images.

- **Force Extra Sorting for All 3D Frames:** Selecting this check box will cause Tecplot to use extra sorting in all 3-D frames.
- **Color to Mono:** This calls up the Monochrome/Gray Scale Mappings dialog. You may examine or modify the mappings from colors used for screen lines or text and floods to gray scales used in monochrome plotting. By default, all lines and text are converted to black. Flood colors are mapped to a gray scale except for custom colors, which are set to black. This option is only available when producing non-color (gray scale) Windows Metafiles.

### 23.2.3. Clipboard Capability for Placing Tecplot Images Directly into Other Applications

Tecplot’s Cut, Copy, and Paste commands work only within Tecplot. However, the Copy Plot to Clipboard command (available only in the Windows version of Tecplot) allows you to copy and paste Tecplot images directly into other applications such as Microsoft Word and Paint, Adobe FrameMaker, and many other applications. The Copy Plot to Clipboard dialog is shown in Figure 23-4.

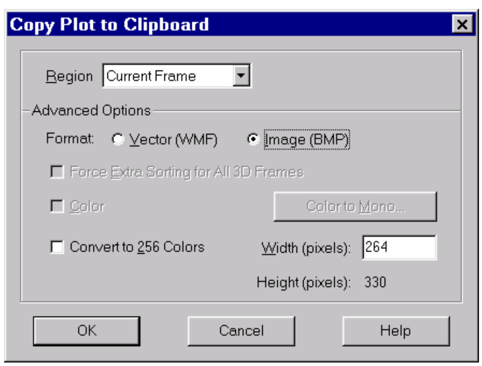



**Figure 23-4.** The Copy Plot to Clipboard dialog (Windows only).

The following options are available:

- **Region:** By default, the image region is the current frame, but you can also choose the smallest rectangle containing all frames, or the full workspace. To specify the image region, choose the desired option (Current Frame, All Frames, or Work Area) from the Image Region drop-down.

- **Format:** Plots may be copied as a vector (Windows Metafile) or image (BMP) format. If you select Image on the Copy Plot to Clipboard dialog you may specify the width in pixels of the bitmap. The greater the width, the greater the quality of your final image.
- **Force Extra Sorting for All 3D Frames:** Selecting this check box will cause Tecplot to use extra sorting in all 3-D frames. This option is only available for Windows Metafile.
- **Color:** Allows you to create color Windows Metafile images.
- **Color to Mono:** This calls up the Monochrome/Gray Scale Mappings dialog. You may examine or modify the mappings from colors used for screen lines or text and floods to gray scales used in monochrome plotting. By default, all lines and text are converted to black. Flood colors are mapped to a gray scale except for custom colors, which are set to black. This option is only available for Windows Metafile in gray scale, not color.
- **Convert to 256 Colors:** If you select this check box, the bitmap will be reduced from up to 16 million potential colors to 256 colors. Tecplot will select the best color match. The converted image will take up less memory on your Windows clipboard.
- **Width:** Select the width of the image in pixels. The image will be scaled to the width you specify. The greater the width you specify, the longer it will take to export the image. However, a larger width will increase the quality of your image. This option is only available for BMP.

To copy and place a Tecplot image into a document or other art work:

1. In the Edit menu, click Copy Plot to Clipboard. Choose options, then click OK.
2. In your other software package, place your cursor or select the frame or site where you want to place the Tecplot image.
3. From the menu of the other software package, execute the Paste command. The entire Tecplot frame will be pasted. Some packages will push other content out of the way to create a spot for the Tecplot image, while others will draw the Tecplot image on top of existing content.
4. Resize and reposition as needed within the other software package.

## 23.3. Creating Image Export Files

In this section, we describe each image export dialog. All export formats allow you to specify the region to be included in the image and the file in which to write the image. Additionally, all export formats allow you to specify the width of the image file in pixels. The exported image will be scaled to this width. Most of the formats have additional options. The file dialog is automatically launched after selecting the export file options and clicking OK.

### 23.3.1. Creating PNG Images

When you select PNG you have the following options:

- **Image Region:** By default, the image region is the current frame, but you can also choose the smallest rectangle containing all frames, or the full workspace. To specify the image region, choose the desired option (Current Frame, All Frames, or Work Area) from the Image Region drop-down.
- **Width:** Select the width of the image in pixels. The image will be scaled to the width you specify. The greater the width you specify, the longer it will take to export the image. However, a larger width will increase the quality of your image.
- **Convert to 256 Colors:** If you select this check box, the bitmap will be reduced from up to 16 million potential colors to 256 colors. Tecplot will select the best color match. The converted image will have a greatly reduced file size.

### 23.3.2. Creating BMP Images

When you select BMP you have the following options:

- **Image Region:** By default, the image region is the current frame, but you can also choose the smallest rectangle containing all frames, or the full workspace. To specify the image region, choose the desired option (Current Frame, All Frames, or Work Area) from the Image Region drop-down.
- **Width:** Select the width of the image in pixels. The image will be scaled to the width you specify. The greater the width you specify, the longer it will take to export the image. However, a larger width will increase the quality of your image.
- **Convert to 256 Colors:** If you select this check box, the bitmap will be reduced from up to 16 million potential colors to 256 colors. Tecplot will select the best color match. The converted image will have a greatly reduced file size.

### 23.3.3. Creating AVI Files

The AVI format is used for viewing movies created in Tecplot. The AVI Exporter dialog is shown in Figure 23-5. Your options are:

- **Image Region:** By default, the image region is the current frame, but you can also choose the smallest rectangle containing all frames, or the full workspace. To specify the image region, choose the desired option (Current Frame, All Frames, or Work Area) from the Image Region drop-down.

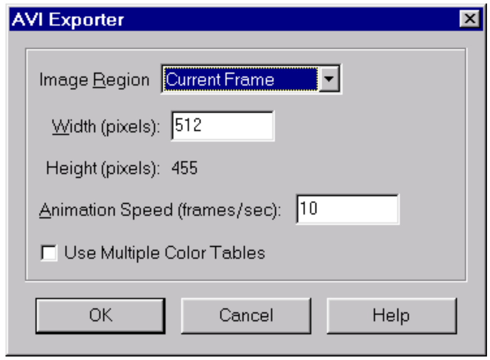



**Figure 23-5.** The AVI Exporter dialog.

- **Width:** Select the width of the image in pixels. The image will be scaled to the width you specify. The greater the width you specify, the longer it will take to export the image. However, a larger width will increase the quality of your image.
- **Animation Speed:** Allows you to set the frames per second. The default speed is ten frames per second. You may only set the animation speed if you are not appending.
- **Use Multiple Color Tables:** If you select this check box, a color table is created for each frame of the animation. If this check box is not selected, Tecplot will scan each frame in your AVI file and create an optimal color table from 256 colors for the entire animation. AVI images are always reduced to 256 colors.

For more information on Audio Visual Interleaved files, see Chapter 30, “Animation and Movies.”

## 23.3.4. Creating TIFF Images

TIFF (Tagged Image File Format) images can be either color or black-and-white. The TIFF Exporter dialog is shown in Figure 23-6.

When you select TIFF you have the following options:

- **Image Region:** By default, the image region is the current frame. You can also choose the smallest rectangle containing all frames, or the full workspace. To specify the image region, choose the desired option (Current Frame, All Frames, or Work Area) from the Image Region drop-down.
- **Width:** Enter a value in the text field for your exported image’s width in pixels. The image region is rendered to the image file to exactly fit a size of Width by Height.

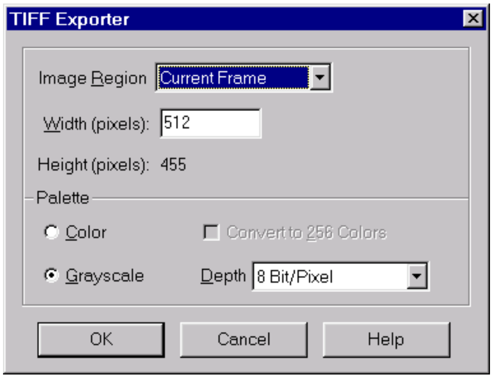



**Figure 23-6.** The TIFF Exporter dialog.

- **Color:** Allows you to create color TIFF images.
- **Gray Scale:** Select this check box to save the image as a gray scale TIFF file.
- **Convert to 256 Colors:** For color images, select this check box to reduce the bitmap from 16 million potential colors to 256 colors. Tecplot will select the best color match. The converted image will have a greatly reduced file size.
- **Depth:** For gray scale images, this specifies the number of shades of gray per pixel. Your options are:
  - **On/Off:** One bit per pixel using an on/off strategy. All background pixels are made white (on), and all foreground pixels, black (off). This setting creates small files and is good for images with lots of background, such as XY-plots and contour lines.
  - **1 Bit/Pixel:** One bit per pixel using gray scale values of pixels to determine black or white. Those pixels that are more than 50 percent gray are black; the rest are white. The larger the number of bits per pixel, the larger the resulting file.This setting creates small files and is good for images with a large amount of foreground and a good deal of contrast (such as contour flooded and light source shaded surfaces).
  - **4 Bit/Pixel:** Four bits per pixel using sixteen levels of gray scale. This is the best setting for TIFF EPS-Preview for programs that do not accept two bits per pixel images. This setting generates larger image files with a fair number of gray levels.
  - **8 Bit/Pixel:** Eight bits per pixel using 256 levels of gray. This setting is useful for full image representation. The files generated by this setting are quite large, and there are so many levels of gray that it is hard to distinguish adjacent ones.

### 23.3.5. Creating Sun Raster Files

Sun Raster files can be created in either of two formats—the standard format, which is not compressed, and a byte-encoded format, which creates a compressed format. The Sun Raster Exporter dialog is shown in Figure 23-7.

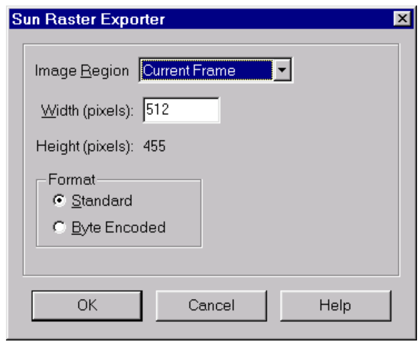



**Figure 23-7.** The Sun Raster Exporter dialog.

The options available for Sun Raster are:

- **Image Region:** By default, the image region is the current frame. You can also choose the smallest rectangle containing all frames, or the full workspace. To specify the image region, choose the desired option (Current Frame, All Frames, or Work Area) from the Image Region drop-down.
- **Width:** Enter a value in the text field for your exported image's width in pixels. The image region is rendered to the image file to exactly fit a size of Width by Height.
- **Format:** You may select Standard, which will create an uncompressed file, or Byte-Encoded, which will create a compressed file. You should select Byte-Encoded unless you have a compelling reason to do otherwise.

### 23.3.6. Creating Raster Metafiles

The Raster Metafile format was defined and is used by NASA, but is also the format read by the Framer program for viewing movies created in Tecplot. The Raster Metafile Exporter dialog is shown in Figure 23-8.

Raster Metafiles and AVI files are the only formats which allow you to put multiple images in a single file. This is useful for creating movies. Your options for Raster Metafiles are:

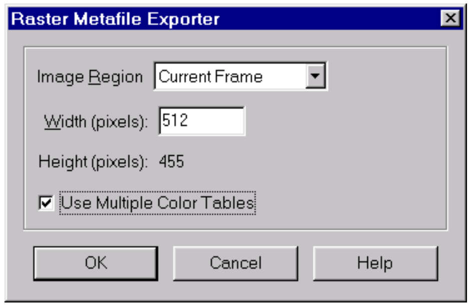



**Figure 23-8.** The Raster Metafile Exporter dialog.

- **Image Region:** By default, the image region is the current frame, but you can also choose the smallest rectangle containing all frames, or the full workspace. To specify the image region, choose the desired option (Current Frame, All Frames, or Work Area) from the Image Region drop-down.
- **Width:** Enter a value in the text field for your exported image's width in pixels. The image region is rendered to the image file to exactly fit a size of Width by Height.
- **Use Multiple Color Tables:** If you select this check box create a color table for each frame of the animation. If this check box is not selected, Tecplot will scan each frame in your AVI file and create an optimal color table from 256 colors for the entire animation. AVI images are always reduced to 256 colors.

For more information on Raster Metafiles and Framer, see Chapter 30, "Animation and Movies."

## 23.3.7. Creating PostScript Images

A PostScript Image file is a raster Encapsulated PostScript file without a preview image—it has advantages and disadvantages compared to PostScript. If your plot uses translucency, or any other features not supported by vector PostScript, you should use a PostScript Image file. If your plot does not use any of the features affected by these limitations, you should probably use a normal PostScript print file or Encapsulated PostScript export file. For complicated plots, PostScript Images can provide much smaller files than vector PostScript. In general, if you want PostScript output, you should create a normal PostScript print file or an Encapsulated PostScript export file.

The PostScript Image Exporter dialog is shown in Figure 23-9.

The following options are available:

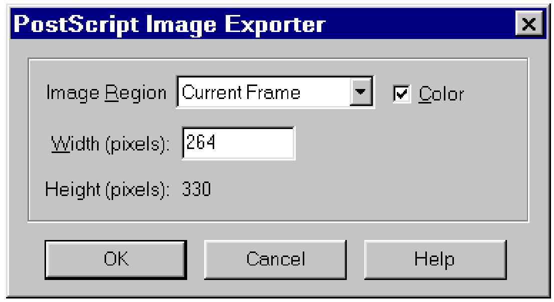



**Figure 23-9.** The PostScript Image Exporter dialog.

- **Image Region:** By default, the image region is the current frame, but you can also choose the smallest rectangle containing all frames, or the full workspace. To specify the image region, choose the desired option (Current Frame, All Frames, or Work Area) from the Image Region drop-down.
- **Color:** Select this check box to save the image as a color PostScript Image file.
- **Width:** Enter a value in the text field for your exported image's width. The image region is rendered to the image file to exactly fit a size of Width by Height.

All color PostScript Image files use a full range of 16 million potential colors. Specifying large widths in PostScript Image files will result in a higher quality image. However, the time it takes to print these files will be greatly increased.

# CHAPTER 24 Data Spreadsheet

All ordered and finite-element data can be viewed using Tecplot's data spreadsheet. The data may be modified within the spreadsheet in order to change the plots Tecplot produces.

The spreadsheet only allows the viewing and altering of data loaded into Tecplot. If you want to add zones, variables, or values, you can do so in your original data files before loading into Tecplot, or through the Create Zone or Alter options on the Data menu.

## 24.1. Viewing a Data Set

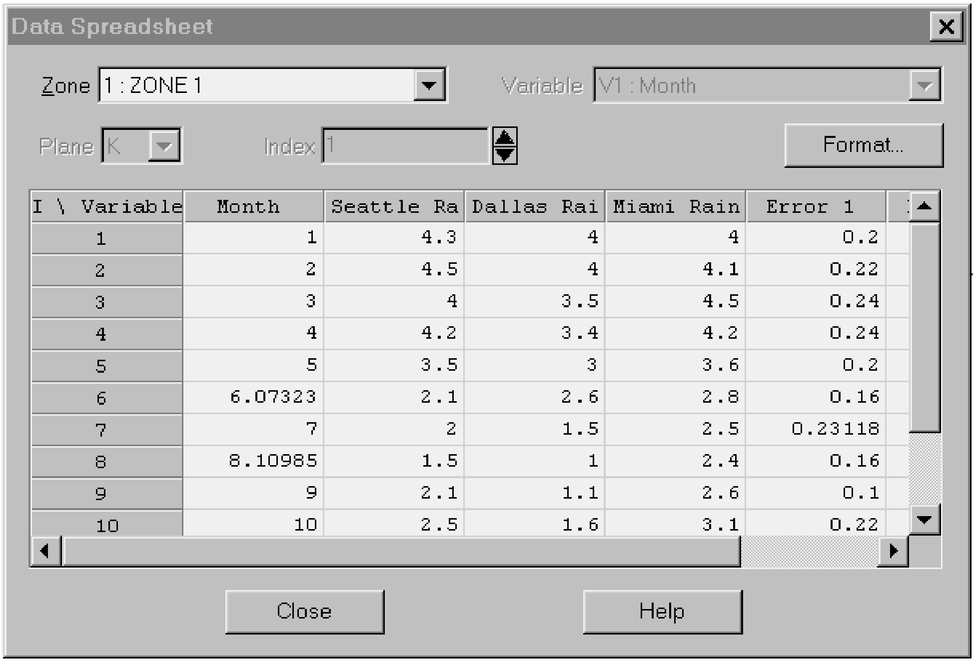
The spreadsheet displays Tecplot's data differently depending on the type of zone being examined. I-ordered and finite element data sets are displayed with each zone's variable displayed in a column. IJ-ordered data sets are displayed in the spreadsheet with I along the rows and J along the columns. IJK-ordered data sets are displayed one plane at a time: selecting the K-plane displays I along the rows and J along the columns, selecting the J-plane displays I along the rows and K along the columns, and selecting the I-plane displays J along the rows and K along the columns. With IJK-ordered data the slice of interest can be selected by entering a specific index or using the up and down arrows provided.

You view data using the Spreadsheet option under the Data menu. To view your data set:

1. From the Data menu, choose Spreadsheet. The Data Spreadsheet dialog appears (see Figure 24-1).
2. From the Data Spreadsheet dialog select a desired zone and variable to examine.
3. Use the scroll bars to examine all of your data.

You can change the format of data in a spreadsheet without changing the appearance of your plot. To change the data spreadsheet's display format:

1. On the Data Spreadsheet dialog click Format. The Data Format dialog appears.
2. Select a number format from the option menu that best represents the data of interest.
3. If available for the selected number format, specify precision (number of decimal places).
4. Enter a column width that best fits the data of interest.




| I \ Variable | Month | Seattle Ra | Dallas Rai | Miami Rain | Error 1 |
|---|---|---|---|---|---|
| 1 | 1 | 4.3 | 4 | 4 | 0.2 |
| 2 | 2 | 4.5 | 4 | 4.1 | 0.22 |
| 3 | 3 | 4 | 3.5 | 4.5 | 0.24 |
| 4 | 4 | 4.2 | 3.4 | 4.2 | 0.24 |
| 5 | 5 | 3.5 | 3 | 3.6 | 0.2 |
| 6 | 6.07323 | 2.1 | 2.6 | 2.8 | 0.16 |
| 7 | 7 | 2 | 1.5 | 2.5 | 0.23118 |
| 8 | 8.10985 | 1.5 | 1 | 2.4 | 0.16 |
| 9 | 9 | 2.1 | 1.1 | 2.6 | 0.1 |
| 10 | 10 | 2.5 | 1.6 | 3.1 | 0.22 |

**Figure 24-1.** The Data Spreadsheet dialog, accessed under the Data menu. The dialog displays the contents of file **`tec90/demo/plt/rain.plt`**.

## 24.2. Changing Data in the Spreadsheet

You can change your data set within Tecplot without changing your original data file. You do this by editing values in the cells of the spreadsheet. To modify data:

1. From the Data Spreadsheet dialog select a desired zone and variable to modify.
2. Select the value of interest from the spreadsheet. This will highlight and expand the value to its full precision.
3. To replace the highlighted value simply enter the new value. Anything highlighted is instantly replaced with new digits entered.
4. To slightly modify a highlighted value select the value a second time. This will un-highlight the value and place the edit cursor at the desired position. Make desired modifications to the existing value.
5. To undo a modification of a given cell press Esc. To commit to a modification press the Enter, Tab, or Shift-Tab keys, or click on another cell.

# CHAPTER 25 Data Operations

All plots in Tecplot, with the exception of sketches, rely on the data sets attached to each frame. You can modify, create, transform, interpolate, duplicate, and delete the data in the current data set using the Data menu. You can also use the data operation capabilities of Tecplot to create plots of analytical functions. By using Tecplot's macro capabilities and equation files, you can create complex data operations that can be repeated on different data sets.

Changes to the data set within Tecplot do not affect the original data file(s). You can save the modified data to a data file by selecting Write Data File from the File menu. When you save a layout file, any data sets that have been modified are also saved to data files (see Section 6.3, "Layout Files, Layout Package Files and Stylesheets," for details).

## 25.1. Altering Data with Equations

Tecplot allows you to alter data in existing zones. You can alter the values of a variable or create new variables as functions of existing variables and index values. Data can be altered simultaneously in one or more zones (or in all zones). For ordered zones, you can also restrict the alteration to specified ranges of indices (I, J, and K).

To modify your data set, follow these steps.

1. From the Data menu, choose Alter.
2. From the Alter menu, choose Specify Equations. The Specify Equations dialog appears as shown in Figure 25-1.
3. Using the Specify Equations dialog, perform the following steps:
   - Enter the equations.
   - (Optional) Select the set of zones to alter using the appropriate region of the Specify Equations dialog. The default is to alter all zones. You may skip this step if you want to apply the equation to all zones.

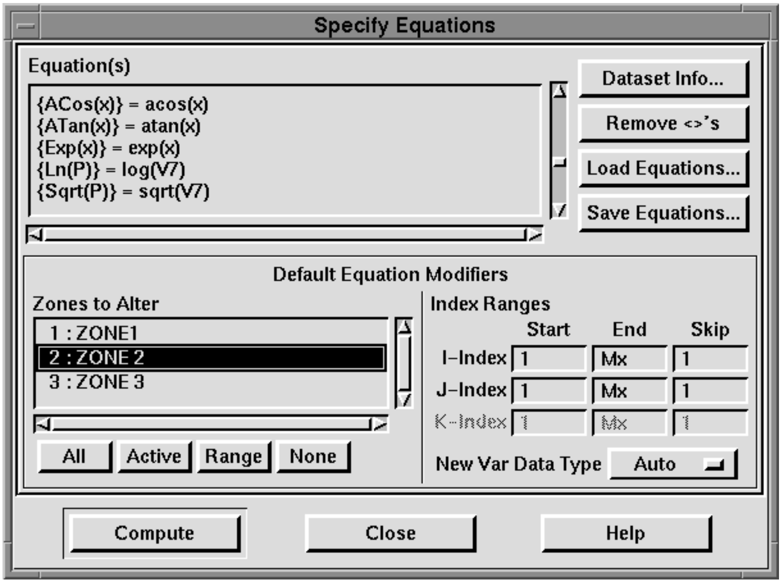



**Figure 25-1.** The Specify Equations dialog.

- (Optional) Select the index ranges to alter in the selected zones. You may skip this step if you want to apply the equation to all points of the selected zones.

4. Click Compute to alter the data. This is important. If you do not click Compute, the data values do not change.

These steps are discussed in more detail in the following sections.

## 25.1.1. Equation Syntax

Tecplot equations have the following form:

```
LValue = F(RValue1, RValue2, RValue3, ...)
```

Where *F()* is a mathematical expression, with some limitations and some extensions as discussed below. *LValue* is a reference to an existing or non-existing variable, and *RValueN* is a reference to a value (such as a constant, variable value, or index value).

There may be any number of spaces within the equation, between operators, and so forth. There cannot, however, be any spaces between the letters of intrinsic-function names nor for variables referred to by name. (See Section 25.1.1.1, “Equation Variables and Values.”)

If the variable on the left-hand side (*LValue*) already exists in the data set of the active frame, the equation is used to modify that variable. If the variable does not already exist, the equation is used to create a new variable as a function of existing variables.

If the equation contains a syntax error, the equation is displayed in an error dialog. The error dialog informs you of the approximate location of the syntax error.

Each equation occupies one line of the Equation(s) text field of the Specify Equations dialog. You can use multiple equations, all of which use as defaults, the zones and index ranges set up at the time you click Compute. Each equation is applied to all specified zones and data points before subsequent equations are computed.

**25.1.1.1. Equation Variables and Values.** A variable is specified in one of three ways: according to its order in the data file, by its name, or by a letter code.

A variable may be referenced according to its order in the data file. **V1** is the first variable in the data file, **V2** is the second, and so forth.

For example, to set the first variable in the data file equal to the sum of the values of the second and third variables, type:

```
V1 = V2 + V3
```

To create a new variable using this specification, you must specify the number of the next available variable. Assuming there are five variables in the data file, you can create a new variable that is equal to half of the fourth variable as follows:

```
V6 = V4 / 2
```

You must specify the number of the next available variable. If you try to assign the result to a higher numbered variable, Tecplot pops up an error dialog informing you that you have specified an invalid variable number.

A variable may also be referenced by its name. You refer to a variable by its name by enclosing the name within curly braces (“**{**” and “**}**”). For example, to set **V3** equal to the value of the variable named **R/RFR**, you can type:

```
V3 = {R/RFR}
```

Variable names are case insensitive, meaning that any combination of uppercase and lowercase letters matches the variable name. Leading and trailing spaces are also not considered. So the following equations are equivalent:

```
v3 = {R/rfr}
V3 = { r/rfr }
```

Spaces within the variable name are significant, so the following equation is not equivalent to the equations above:

```
V3 = {R / rfr}
```

If two or more variables have the same name, Tecplot uses the first variable. So, if both **`V5`** and **`V9`** are named **`R/rfr`**, **`V5`** is used.

The curly braces can also be used on the left-hand side of the equation. In this case, if a variable with that name does not exist, a new variable is created with that name. This is useful in equation files (see Section 25.1.1.6, “Examples of Equations,” for details). For example, the following equation sets a variable called **`T/R`** to the value of a variable called **`T`** divided by the value of a variable called **`R`**. If no variable called **`T/R`** exists, a new variable is created.

```
{T/R} = {T} / {R}
```

Finally, variables may also be referenced by letter codes. Letter codes can also be used to reference index values. Valid letter codes are:

- **`I`**: The I-index value at the data point.
- **`J`**: The J-index value at the data point (1 for finite-element zones).
- **`K`**: The K-index value at the data point (1 for finite-element zones).
- **`X`**: The variable assigned to the X-axis (in XY-plots, all active mappings must have the same X variable in order for this variable name to be valid).
- **`Y`**: The variable assigned to the Y-axis (in XY-plots, all active mappings must have the same Y variable in order for this variable name to be valid).
- **`Z`**: The variable assigned to the Z-axis (if in 3-D).
- **`U`**: The X-component of vectors (if set).
- **`V`**: The Y-component of vectors (if set).
- **`W`**: The Z-component of vectors (if set, and if in 3-D).
- **`B`**: The value-blanking variable (if set).
- **`C`**: The contour variable (if set).
- **`S`**: The scatter-sizing variable (if set).

Letter codes may be used anywhere on the right-hand side of the equation. Do not enclose them in curly braces. Some examples follow:

```
V3 = I + J
V4 = cos(X) * cos(Y) * cos(Z)
```

```
{Dist} = sqrt(U*U + V*V + W*W)
{temp} = min(B,1)
```

Those letter codes representing variables (all letter codes except **I**, **J**, and **K**) may be used on the left-hand side of the equation, as well. For example:

```
Z = X*X/(1+Y*Y)
W = 0
S = 1+ABS(S)
```

You receive an error message if the appropriate Tecplot feature is not available. For example, if you try to use **Z** and the current frame is not in 3D frame mode, you get an error message. The variables referenced by the letter codes are for the current frame.

**25.1.1.2. Data Set Information.** To view a list of variable names and numbers, select Data Set Info on the Data menu, or click Data Set Info on the Specify Equations dialog. The Data Set Information dialog appears, as shown in Figure 25-2, listing all the variables in the current data set.

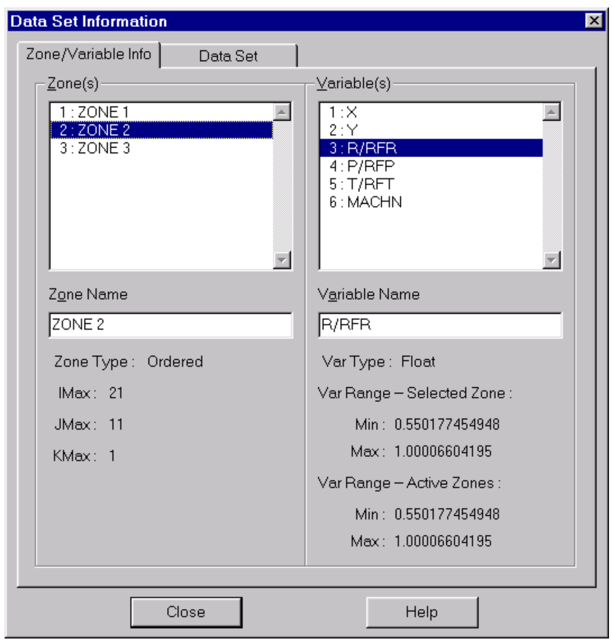



**Figure 25-2.** The Data Set Information dialog.

To see more information about a zone, select one of the zone names listed under Zone(s) in the upper left-hand corner. The information about that zone will be displayed in the lower left-hand corner. If it is an ordered zone, the I-, J-, and K-values will be shown. If it is a finite-element zone, the number of points and elements will be shown. While a zone is selected you may edit its name in the Zone Name text field.

To view more information about a variable, select one of the variable names listed under Variables in the upper right-hand corner. Information about that variable, such as whether it is float or integer, its range in values, and the range in values for all active zones, will be shown in the lower right-hand corner. To see a variable's range for a particular zone, select the zone from Zone(s) while the variable is highlighted under Variables. While a variable is selected you may edit its name in the Variable Name text field.

To view additional information about the data set click on the Data Set tab. If the data has not been changed since a data file was loaded, the name of that data file will appear in the Data File(s) field. The Data Set Title field shows the current name of the data set. While a data set is selected you may edit its name in this field. The lower part of the page shows the Variable Load Mode, which will affect your ability to load additional data files to append your current data set. See Chapter 5 for a discussion of loading variables by name or position. The Locked By field will inform you if the current data set has been locked by an add-on. Add-ons can lock a data set which in turn prevents your from deleting zones or deleting the last frame associated with the data set.

**25.1.1.3. Equation Operators and Functions.** In an equation, the valid binary operators are as follows:

| | |
|---|---|
| `+` | Addition. |
| `-` | Subtraction. |
| `*` | Multiplication. |
| `/` | Division. |
| `**` | Exponentiation. |

Binary operators have the following precedence:

| | |
|---|---|
| `**` | Highest precedence. |
| `*,/` | |
| `+,-` | Lowest precedence. |

For example, the expression **`V1+2*V2`** is evaluated as **`V1+(2*V2)`**, not as **`(V1+2)*V2`**. Operators are evaluated from left to right within a precedence level (that is, "*" and "/" are evaluated left to right).

The available functions are as follows (except where noted, all take a single argument):

| | |
|---|---|
| **SIN** | Sine (angle must be specified in radians). |
| **COS** | Cosine (angle must be specified in radians). |
| **TAN** | Tangent (angle must be specified in radians). |
| **ABS** | Absolute value. |
| **ASIN** | Arcsine (result is given in radians). |
| **ACOS** | Arccosine (result is given in radians). |
| **ATAN** | Arctangent (result is given in radians). |
| **ATAN2(A,B)** | Arctangent of **A/B** (result is given in radians). |
| **SQRT** | Positive square root. |
| **LOG, ALOG** | Natural logarithm (base *e*). |
| **LOG10, ALOG10** | Logarithm base 10. |
| **EXP** | Exponentiation (base *e*); **EXP(V1)=*e***(V1)**. |
| **MIN(A,B)** | Minimum of **A** or **B**. |
| **MAX(A,B)** | Maximum of **A** or **B**. |
| **SIGN** | Returns -1 if argument is negative, +1 otherwise. |
| **ROUND** | Round off to the nearest integer. |
| **TRUNC** | Remove fraction part of a value. |

**LOG** and **ALOG** are equivalent functions, as are **LOG10** and **ALOG10**.

First- and second-derivative and difference functions are also available. These are discussed later in this section.

To call an intrinsic function, place its argument within parentheses ("**(**" and "**)**"). For example, to set **V4** to the arctangent of **V1**, use:

```
V4 = ATAN(V1)
```

**25.1.1.4. Specifying Zone Numbers.** By following a variable reference with square brackets ("**[**" and "**]**"), you can specify a specific zone from which to get the variable value. Some examples follow:

```
V3 = V3 - V3[1]
X = ( X[1] + X[2] + X[3] ) / 3
{TempAdj} = {Temp}[7] - {Adj}
V8 = V1[19] - 2*C[21] + {R/T}[18]
```

The zone number must be a positive integer constant less than or equal to the number of zones. The zone specified must have the same structure (I-, IJ-, or IJK-ordered or finite-element) and dimensions (*IMax*, number of nodes) as the current zone. If you do not specify a zone, the current zone is used.

Zone specification works only on the right-hand side of the equation. All values used on the right-hand side of the equation are the values before the alteration began. To specify zones for the left-hand side, use the Zones region of the Specify Equations dialog.

**25.1.1.5. Specifying Indices.** By following a variable reference with parentheses ("**(**" and "**)**"), you can specify indices. Indices can be absolute or an offset from the current index. For example:

```
V2 = ( V2(i+1) + V2(i-1) ) / 2
U = U(i+1,j) - U(i-1,j) + V(i+2,1) + 3*W(i-1)
{NTQ} = {TQ} + {TQ}(i-3,j+7,k-1) - {TQ}(3,j+1,k+8}
S = S(i+1,j) - V3(2) + {RFR/T}(J+2)
```

Index offsets are specified by using the appropriate index "**i**", "**j**" or "**k**" followed by a "**+**" or "**-**" and then an integer constant. Any integer offsets may be used. If the offset moves beyond the end of the zone, the boundary value is used. For example, **V3(i+2)** uses the value **V3(IMAX)** when *I=IMax-1* and *I=IMax*. **V3(I-2)** uses the value of **V3(1)** when*I=1* or *I=2*.

Absolute indices are specified by using a positive integer constant only. For example, **V3(2)** references **V3** at index **2** regardless of the current **i** index.

Indices must be listed I-index, then J-index, then K-index. The J-index is omitted if the data set is I-ordered; the K-index is omitted if the data set is not IJK-ordered. Indices are not allowed for finite-element data.

Index specification works only on the right-hand side of the equation. If the indices are not specified, the current index values are used. To specify indices for the left-hand side, use the Index Ranges section of the Specify Equations dialog.

Indices may be combined with zone specifications. The zone is listed first, then the index offset. For example:

```
V3 = V3 - V3[1](i+1)
Y = Y[1] - Y[2](1) + Y(1,j+3) + Y
```

**25.1.1.6. Examples of Equations.** In the following equation, **V1** (the first variable defined in the data file) is replaced by two and a half times the existing value of **V1**:

```
V1 = 2.5*V1
```

The following equation sets the value of a variable called **Density** to 205. A new variable is created if a variable called **Density** does not exist.

```
{Density} = 205
```

In the next equation, the values for **Y** (the variable assigned to the Y-axis) are replaced by the negative of the square of the values of **X** (the variable assigned to the X-axis):

```
Y = -X**2
```

The following equation replaces the values of **V3** with the values of **V2** rounded off to the nearest integer. A new variable is created if there are only two variables currently in the data set.

```
V3 = round(V2)
```

In the following equation, the values of the fourth variable in the data set are replaced by the log (base 10) of the values of the third variable.

```
V4 = ALOG10(V3)
```

Suppose that the third variable and fourth variable are the X- and Y-components of velocity and that there are currently a total of five variables. The following examples create a new variable (**V6**) that is the magnitude of the components of velocity.

```
V6 = (V3*V3+V4*V4)**0.5
```

or

```
V6 = sqrt(V3**2+V4**2)
```

The above operation can also be accomplished with the following equation (assuming you have already defined the vector components for the current frame):

```
{Mag} = sqrt(U*U + V*V)
```

The following equation sets the value of a variable named **diff** to the truncated value of a variable named **depth** subtracted from the existing value of **depth**:

```
{diff} = {depth} - trunc({depth})
```

In the next equation, **C** (the contour variable) is set to the absolute value of **S** (the scatter-sizing variable), assuming both **C** and **S** are defined:

```
C = abs(S)
```

In the following example, a new variable is created (assuming that only seven variables initially existed in the data file). The value for **V8** (the new variable) is calculated from a function of the existing variables:

```
V8 = SQRT((V1*V1+V2*V2+V3*V3)/(287.0*V4*V6))
```

The above operation could have been performed in two simpler steps as follows:

```
V8 = V1*V1+V2*V2+V3*V3
V8 = SQRT(V8/(287.0*V4*V6))
```

The following equation replaces any value of a variable called **TIME** that is below **5.0** with **5.0**. In other words, the values of **TIME** are replaced with the maximum of the current value of **TIME** and **5.0**:

```
{TIME} = max({TIME},5)
```

The following equation creates variable **V4** which has values of **X** at points where **X**<0; at other points, it has a value of zero (this does not affect any values of **X**):

```
V4 = min(X,0)
```

Another example using intrinsic functions is shown below:

```
V8 = 55.0*SIN(V3*3.14/180.0) + ALOG(V4**3/(v1+1.0))
```

You can also reference the I-, J-, and K-indices in an equation. For example, if you wanted to cut out a section of a zone using value-blanking, you could create a new variable that is a function of the I- and J-indices (for IJ-ordered data). Then, by using value-blanking, you could remove certain cells where the value of the value-blanking variable was less than or equal to the value-blanking cut-off value.

Here is an example for calculating a value-blanking variable that is zero in a block of cells from *I=10* to 30, and is equal to one in the other cells:

```
V3 = min((max(I,30)-min(I,10)-20),1)
```

The following equation replaces all values of **Y** with the difference between the current value of **Y** and the value of **Y** in zone 1. (If zone 1 is used for the data alteration, the new values of **Y** will be zero throughout that zone.)

```
Y = Y - Y[1]
```

The following equation replaces the values of **V3** (in an IJ-ordered zone) with the average of the values of **V3** at the four adjacent data points:

```
V3 = (V3(i+1,j)+V3(i-1,j)+V3(i,j+1)+V3(i,j-1))/4
```

The following equation sets the values of a variable called **TEMP** to the product of the values of a variable called **T** measured in four places: in zone 1 at two index values before the current data point, in the current zone at an absolute index of three, in zone 4 at the current data point, and in the current zone at the current data point.

```
{TEMP} = {T}[1](i-2) * {T}(3) * {T}[4] * {T}
```

**25.1.1.7. Derivative and Difference Functions.** Tecplot has a complete set of first- and second-derivative and difference functions. These functions are listed below:

```
ddx     ddy     ddz
d2dx2   d2dy2   d2dz2
d2dxy   d2dyz   d2dxz
ddi     ddj     ddk
d2di2   d2dj2   d2dk2
d2dij   d2djk   d2dik
```

The derivative function **ddx** is used to calculate $\partial/\partial x$; **d2dx2** calculates $\frac{\partial^2}{\partial x^2}$ ; **d2dxy** calculates $\frac{\partial^2}{\partial x \partial y}$, and so on. The difference functions **ddi**, **d2di2**, etc., calculate centered differences of their argument with respect to the indices I, J, and K based on the indices of the point. For example:

$$\mathbf{ddi}(V) = \frac{V_{i+1} - V_{i-1}}{2}$$

The derivative and difference functions are used just like the intrinsic functions described above. For example:

```
V4 = ddx(V3)
V6 = d2dx2(v5)
{dC/dx} = ddx(C)
V8 = ddj(X)
{Vt12} = ddy({Vt11}(i+1)) + ddy({Vt11}(i-1))
Z = d2dj2(sin(v5*v6))
V9 = ddj(ddx({R/T}))
```

```
C = d2dij(C[1]-C)
{NEWVAR}=ddi(X)+ddj(Y)+ddk(Z)
```

The use of derivative and difference functions is restricted as follows:

- Finite-element zones are not supported. Derivative and difference functions can be used only on I-, IJ-, and IJK-ordered zones.
- Derivatives and differences for IJK-ordered zones are calculated for the full 3-D volume. The IJK-mode for such zones is not considered.
- If the derivative cannot be defined at every data point in all the selected zones, the operation is not performed for any data point.
- Derivative functions are calculated using the current frame's axis assignments. Be careful if you have multiple frames with different variable assignments for the same data set.
- All derivatives and differences are centered at the data point.
- Derivatives at the boundary of two zones may differ since Tecplot operates on only one zone at a time while generating derivatives.

Boundary values for first-derivative and difference functions (**ddx**, **ddy**, **ddz**, **ddi**, **ddj**, and **ddk**) are evaluated in one of two methods: simple or complex. The default is simple. The following parameter in the configuration file selects the method to use:

```
$!INTERFACE
  DATA {DERIVATIVEBOUNDARY=SIMPLE}
```

Change the parameter **SIMPLE** to **COMPLEX** to use the complex boundary condition.

For simple boundary conditions, the boundary derivative is determined by the one-sided first derivative at the boundary. This is the same as assuming that the first derivative is constant across the boundary (with the second derivative equal to zero).

For complex boundary conditions, the boundary derivative is extrapolated linearly from the derivatives at neighboring interior points. This is the same as assuming that the second derivative is constant across the boundary (with the first derivative varying linearly).

For second-derivatives and differences (**d2dx2**, **d2dy2**, **d2dz2**, **d2dxy**, **d2dyz**, **d2dxz**, **d2di2**, **d2dj2**, **d2dij**, **d2dk2**, **d2djk**, and **d2dik**), these boundary conditions are ignored. The boundary derivative is set equal to the derivative one index in from the boundary. This is the same as assuming that the second derivative is constant across the boundary.

You can create your own derivative boundary conditions by using the index range and the indices options discussed previously.

### 25.1.2. Zone Selection

You may select which zone(s) to alter using the Zones to Alter area of the Specify Equations dialog. By default, all zones are altered. If you are creating a new variable, all zones must be selected since all zones in a data set must have the same number of variables per data point.

### 25.1.3. Index Range and Skip Selections for Ordered Zones

For ordered zones, you can use the Index Ranges area of the Specify Equations dialog to select which data points to alter (based upon the index values). You can specify an I-Index range, a J-Index range, and/or a K-Index range.

For each index range, you enter the start index value, the end index value, and a skip factor. A skip factor of one applies your equation to every index value; two does every other; three, every third; and so forth. Use the special value **`0`** or **`Mx`** to specify the maximum index. You can also use the values **`Mx-1`** (to specify the index one less than the maximum index), **`Mx-2`**, and so forth. By default, these ranges are set to the entire range of points (that is, the start index defaults to one, the end index defaults to **`Mx`**, and the skip factor defaults to one).

The index ranges are applied to all ordered zones that are selected using the Zones to Alter area. Index ranges are ignored for finite-element zones; every data point of a finite-element zone is altered regardless of the settings in the Index Ranges area.

If you are creating a new variable, the new variable’s value is set to zero at any index value that is skipped.

### 25.1.4. Specifying the Data Type for New Variables

If your equations are creating one or more new variables, you can specify a data type for these new variables using the New Var Data Type drop-down menu. By default, this is set to Auto, and Tecplot assigns the most appropriate data type to the variables. However, if you want to be sure that your new variables have a particular data type, choose the type from the New Var Data Type drop-down menu. The following data types are available:

- **Single:** Four-byte floating point values.
- **Double:** Eight-byte floating point values.
- **Long Int:** Four-byte integer values.
- **Short Int:** Two-byte integer values.
- **Byte:** One-byte integer values (zero to 255).
- **Bit:** Either zero or one.

### 25.1.5. Overriding Equation Restrictions

The zone and index restrictions specified in the equation dialog can be overridden on an equation by equation basis. To specify restrictions for a single equation add the colon character (**:**) at the end of the equation followed by one or more of the following:

| Equation Restrictor | Comments |
|---|---|
| <***Z***=<*set*>> | Restrict the zones. |
| <***I***=*start*[,*end*[,*skip*]]> | Restrict the I-range. |
| <***J***=*start*[,*end*[,*skip*]]> | Restrict the J-range. |
| <***K***=*start*[,*end*[,*skip*]]> | Restrict the K-range. |
| <***D***=<*datatype*>> | Set the data type for the variable on the left hand side. This only applies if a new variable is being created. |

For example, to add one to X in zones 1, 3, 4, and 5:

```
X=X+1:<Z=[1,3-5]>
```

The following example adds one to X for every other I-index. Note that zero represents the maximum index.

```
X=X+1:<I=1,0,2>
```

The next example creates a new variable of type Byte:

```
{NewV}=X-Y:<D=Byte>
```

### 25.1.6. Performing the Alteration

After you have set the index ranges and zones that are to be altered and have entered the equations, click the Compute button to apply the equations to your data. Each equation is applied to each data point in each zone independently of the other points. Each equation is applied to all specified zones and data points before subsequent equations are processed.

If an error occurs during the alteration (because of division by zero, overflow, underflow, and so forth), an error message is displayed, and all of the zones are restored to the state they were in before the bad equation was processed. Thus, if you have three equations A, B, and C, and B contains an error, the final state is the result of processing equation A.

### 25.1.7. Equations in Macros

Tecplot allows you to put your equations in macros. In fact, we sometimes refer to a macro with just equations as an equation file. An equation in a macro file is specified using the **$!ALTERDATA** macro command. Equation files may also include comment lines, and in fact must start with the comment **#!MC 900**, just like other macro files. If you are performing complex operations on your data, and/or the operations are repeated frequently, equation files can be very helpful.

You can create equation files from scratch using an ASCII text editor, or you can create your equations interactively using the Specify Equations dialog, and then save the resulting equations. The standard file name extension for equation files is "**.eqn**."

For example, you might define an equation to compute the magnitude of a 3-D vector. In the Specify Equations dialog, it would have the following form:

```
{Mag} = sqrt(U*U + V*V + W*W)
```

In a macro file, it would have the following form:

```
#!MC 900
$!ALTERDATA
  EQUATION = "{Mag} = sqrt(U*U + V*V + W*W)"
```

That is, the interactive form of the equation must be enclosed in double quotes and supplied as a value to the **EQUATION** parameter of the **$!ALTERDATA** macro command.

To read an equation file, click Load Equations on the Specify Equations dialog. The Load Equation File dialog appears. Select an equation file that contains a set of equations to apply to the selected zones of your data. The equations in the equation file are added to the list of equations in the dialog. All equations are applied to your data when you click Compute.

Equations in equation files may be calculated somewhat differently depending on whether the computation is done from within the Specify Equations dialog or by running the equation file as a macro. When loaded into the Specify Equations dialog, equations that do not contain zone or index restrictions use the current zone and index restrictions shown in the dialog. When processed as a macro file, such equations apply to all zones and data points. To include zone and index restrictions, you must include them in the equation file as part of the **$!ALTERDATA** command. See Section 25.1.7.2, "Specifying Zones To Operate On," and Section 25.1.7.4, "Specifying Ranges and Skip Factors," for details.

**25.1.7.1. Equation File Comments.** As with any macro file, any line with a pound sign ("**#**") as the first character is considered a comment and ignored. For example:

```
#!MC 900
#
# calculate magnitudes of vectors
#
$!ALTERDATA
  EQUATION = "{Mag} = sqrt(U*U + V*V + W*W)"
#
# normalize
#
$!ALTERDATA
  EQUATION = "U = U/{Mag}"
$!ALTERDATA
  EQUATION = "V = V/{Mag}"
$!ALTERDATA
  EQUATION = "W = W/{Mag}"
#
# done
#
```

**25.1.7.2. Specifying Zones To Operate On.** To restrict the zones to operate on, specify the number of the zone or zones to the **$!ALTERDATA** command as a square-bracketed parameter:

```
$!ALTERDATA [zonelist]
  EQUATION = "equation"
```

where *zonelist* is a list of zones or zone ranges separated by a comma ("**,**"). Zone ranges are separated by a hyphen ("**-**"). For example, the following macro command restricts the equation to only the second zone:

```
$!ALTERDATA [2]
  EQUATION = "X=X+1"
```

If you do not specify the zones in your equation file, Tecplot defaults to the dialog settings if you read the equations into the Specify Equations dialog. (If you run the equations from a macro file, Tecplot defaults to all zones.) The default setting in the Specify Equations dialog is all zones. To create a new variable, you must have all zones available for data operation.

For example:

```
#!MC 900
# do the following to default set of zones
$!ALTERDATA
  EQUATION = "Y = Y*2"
```

```
# now just do zone 1
$!ALTERDATA [1]
  EQUATION = "Y = 0"
# now just do zones 2 and 3
$!ALTERDATA [2-3]
  EQUATION = "Y = Y[4]"
# now just do zones 4, 5, 6, 7, and 19
$!ALTERDATA [4-7,19]
  EQUATION = "Y = Y(i-1) + Y(i+1)"
# now do all zones to create a new variable
$!ALTERDATA
  EQUATION = "{SomeNewVariable} = 0"
```

**25.1.7.3. Specifying Zone Numbers for Operands.** By following a variable reference with brackets "[" and "]" you may designate a specific zone from which to get the variable value. For example:

```
V3 = V3 -V3[1]
X = ( X[1] + X[2] + X[3] ) / 3
{TempAdj} = {Temp}[7] - {Adj}
V7 = V1[19] - 2*C[21] + {R/T}[18]
```

The zone number must be a positive integer constant less than or equal to the number of zones. The zone designated must have the same structure (finite-element, I-, IJ-, or IJK-ordered) and dimensions (number of nodes and so forth). If you do not designate a zone, the current zone will be used.

Specifying a zone only works on the right-hand side of an equation. All values used on the right-hand side of the equation are the values from before the alteration began.

**25.1.7.4. Specifying Ranges and Skip Factors.** You can restrict index ranges from within your Equation file by specifying the following parameters to the **$!ALTERDATA** macro command:

```
IRANGE = min, max, skip
JRANGE = min, max, skip
KRANGE = min, max, skip
```

where *min* is the minimum value for that index; *max*, the maximum; and *skip*, the skip factor. Index ranges are described in Section25.1.3, "Index Range and Skip Selections for Ordered Zones." If these commands are not used, the default index ranges use the currently selected indices (if the equations are executed from a macro, all points are used). Index ranges are effective only for ordered zones.

For example:

```
#!MC 900
#
# use default index range
#
$!ALTERDATA
  EQUATION = "{K/10} = round(K/10)"
#
# calculate d2dxy(C) everywhere
#
$!ALTERDATA
  IRange = 1,0,1
  JRange = 1,0,1
  KRange = 1,0,1
  EQUATION = "V5 = d2dxy(C)"
#
# set corner values to zero
#
$!ALTERDATA
 IRange = 1,0,0
 JRange = 1,0,0
 KRange = 1,0,0
 EQUATION = "V5 = 0"
```

**25.1.7.5. Specifying New Variable Data Type.** You can specify the data type for new variables that are created in equations in the **$!ALTERDATA** command by including the following parameter:

**DATATYPE = <***datatype***>**

where *datatype* can be one of **SINGLE**, **DOUBLE**, **LONGINT**, **SHORTINT**, **BYTE**, or **BIT**.

For example, to create a new variable and set its data type to **BYTE**:

```
$!ALTERDATA
  EQUATION = "{NewV} = X-Y"
  DATATYPE = BYTE
```

**25.1.7.6. Naming Variables in an Equation File.** To name a variable when creating it in an equation file, simply provide the name within curly braces (**{}**) on the left hand side of the equation, as in the example below:

```
#!MC 900
$!ALTERDATA
  EQUATION = "{myvar} = V4 / V5"
```

To rename a variable using a macro, use the macro command **`$!RENAMEDATASETVAR`**, as in the example below:

```
#!MC 900
$!RENAMEDATASETVAR
  VAR = 6
  NAME = "myvar"
```

The macro listed above may be played back using Play option of the Macro command under the File menu. It cannot, however, be loaded in as an equation in the Specify Equations dialog.

**25.1.7.7. Example Equation Files.** If you want to cut out a section of a zone using value-blanking, you can create a new variable that is a function of the I- and J-indices (for IJ-ordered data). Then, by using value-blanking, you can remove certain cells where the value of the value-blanking variable is less than or equal to the value-blanking cut-off value. Below is an example equation file for calculating a value-blanking variable that is zero in a block of cells from *I=10* to 28 and from *J=5* to 16 and is equal to one in the other cells.

```
#!MC 900
## create a mask for the cells in the I-direction
$!ALTERDATA
  EQUATION =  "{MASK_I} = max(I,28) - min(I,10) - 18"
$!ALTERDATA
  EQUATION = "{MASK_I} = min({MASK_I},1)"
## create a mask for the J-direction
$!ALTERDATA
  EQUATION = "{MASK_J} = max(J,16) - min(J,5) - 11"
$!ALTERDATA
  EQUATION = "{MASK_J} = min({MASK_J},1)"
## create the value-blanking variable that is the
## intersection ("and") of the above conditions
$!ALTERDATA
  EQUATION = "{VBLANK} = {MASK_I} + {MASK_J}"
## create a second value-blanking variable that is
## the union ("or") of the above conditions
$!ALTERDATA
  EQUATION = "{OR_VBLANK} = {MASK_I} * {MASK_J}"
```

Another example of an equation file is shown below. In this example, **`V8`** is a new variable, so all zones must be selected when **`V8`** is created.

```
#!MC 900
## The first three equations are applied to all zones
$!ALTERDATA
  EQUATION = "V4 = SQRT(V4) / 460.0"
$!ALTERDATA
  EQUATION = "V4 = (V4*V5 + V4*V6)/2"
$!ALTERDATA
  EQUATION = "V5 = SIN(V4/V3) * EXP(-0.53*V2/V1)"
## Creating New Variable, Operate on all zones
$!ALTERDATA [1,|NUMZONES|]
  EQUATION = "{CRITT} = 0.5396E-3 * V4**2"
## Operate on Zone 5 only
$!ALTERDATA [5]
  EQUATION = "V1 = V1 / 2"
$!ALTERDATA [5]
  EQUATION = "V2 = V2 / 2"
## Operate On Zones 6,7,8,9, and 23 Only
$!ALTERDATA [6-9,23]
  EQUATION = "V1 = V1 * 4.3"
$!ALTERDATA [6-9,23]
  EQUATION = "V2 = V2 * 4.3"
## Operate On All Zones, But Limit Index Ranges
## To Every Fifth Point in the I- or J-Direction
$!ALTERDATA
  IRange = 1,0,5
  JRange = 1,0,5
  EQUATION = "V5 = EXP(V3(i-2)+V5(i+4))"
##
##
```

## 25.2. Transforming 2-D Polar Coordinates to Rectangular

All 2-D Tecplot plots use a rectangular coordinate system with axes X and Y. Some 2-D data may be represented in polar coordinate form, in which each point is represented by the radius *r* and by an angle $\theta$ (in radians). Such a data set will initially display as a rectangle in Tecplot. Use the Transform Polar to Rectangular dialog to transform data sets in polar coordinates to a rectangular coordinate system which Tecplot can use. When you use this dialog, Tecplot assumes the current X-variable represents the radius *r*, and the current Y-variable represents the angle $\theta$. These variables are listed in the upper portion of the dialog.

To transform your data in polar coordinates to rectangular coordinates:

1. From the Data menu, choose Alter, then choose Polar to Rectangular. The Transform Polar to Rectangular dialog appears as shown in Figure 25-3.

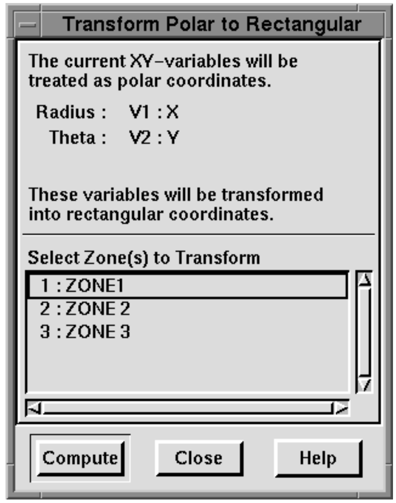



**Figure 25-3.** The Transform Polar to Rectangular dialog.

2. Verify that the X- and Y-variables shown represent *r* and θ, respectively, in your polar coordinate data. The data must be in radians.
3. Select the zones for the transformation.
4. Click Compute.

Redraw you plot to see your data appearing in its transformed state.

## 25.3. Transforming 3-D Spherical Coordinates to Rectangular

This process is the same as the 2-D case discussed in the previous section except that in this case Tecplot assumes the current X-variable represents the radius ρ, the current Y-variable the angle θ (in radians), and the current Z-variable the angle ψ. Figure 25-4 shows ρ, θ, and ψ in the spherical coordinate system.

## 25.4. Rotating 2-D Data

Use the 2D Rotate dialog to rotate 2-D field data about a user specified XY-origin. Unlike 3-D rotation, which does not alter the data but merely the user's view of the data, 2-D rotation actually modifies the data.

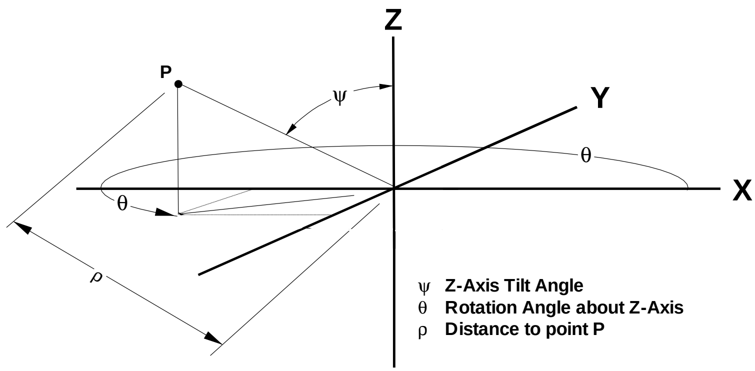



**Figure 25-4.** Three-dimensional angles of rotation.

To rotate data in 2-D:

1. From the Data menu, choose Alter, then choose 2D Rotate. The 2D Rotate dialog appears, as shown in Figure 25-5.
2. Specify the angle of rotation, in degrees, using the Angle (deg) drop-down.
3. Enter the X- and Y-coordinates of the origin of rotation (that is, the point around which the data rotate).
4. Select the zones to rotate.
5. Click Compute.

Redraw your plot to view the rotated data.

## 25.5. Shift Cell-Centered Data

Tecplot works with node-centered data. That is, it assumes that data values are taken at the grid points. Sometimes you may have cell-centered data in which the variable values are specified at the center of cells defined by the mesh grid points. Use the Shift Cell-Centered Data dialog to recalculate the values of variables at your grid points under the assumption that the original data represented values observed at the centers of the grid cells.

For example, suppose you have grid points at *X=1, 2, ...,* and *Y=1, 2, ...,* and your data values are gathered at the cell centers, that is, at *X=1.5, 2.5, ..., Y=1.5, 2.5, ...,* when you create your Tecplot data file, simply map the cell-centered values to the lowest indexed corner of the cell, so that, for example, the values at *(X, Y)=(1.5, 1.5)* are identified with the grid point *(X, Y)=(1,*

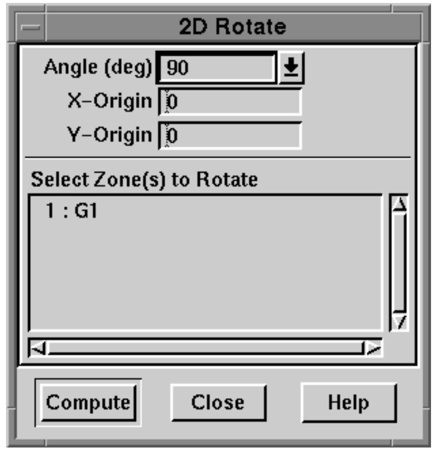



**Figure 25-5.** The 2D Rotate dialog.

*1)*. Then use the Shift Cell-Centered Data dialog to interpolate these cell-centered values onto the grid points. The final result is a node-centered data set with interpolated values at each node. Note that the values supplied along the outer boundary (*I=IMax* or *J=JMax* or *K=KMax*) are not used.

To transform your data to cell-centered:

1. Create your data file as described above.
2. From the Data menu, choose Alter, then choose Shift Cell-Centered Data. The Shift Cell-Centered Data dialog appears as shown in Figure 25-6.
3. Select the zones and variables to be shifted.
4. Click Compute.

## 25.6. Creating Zones

You create zones in Tecplot as part of any number of operations: zone duplication, iso-surface extraction, streamtrace zone generation, 3-D data slicing, finite-element boundary extraction, and triangulation. Most of these tasks are discussed in the chapter related to those topics—streamtrace zone generation, for example, is discussed in Chapter 13, “Streamtraces.” This section concentrates on zone creation that is also essentially data creation—creating new rectangular, circular, or linear zones. We also describe briefly the data duplication aspects of zone creation—zone duplication, mirror zones, and sub-zones.

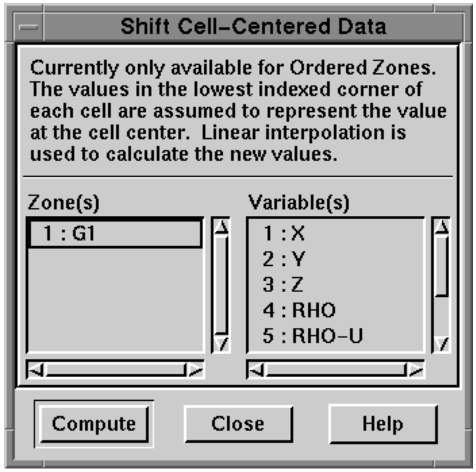



**Figure 25-6.** The Shift Cell-Centered Data dialog.

### 25.6.1. Creating a 1-D Line Zone

A 1D-line zone is an I-ordered set of points along a line. You can create a 1-D line zone as the first step in plotting an analytic function. Typically, you create the 1-D line zone, then use the Specify Equations dialog to modify the Y-variable, then plot the result.

To create the 1-D line zone:

1. From the Data menu, choose Create Zone, then choose 1-D Line. The Create 1-D Line Zone dialog appears.
2. In the text field labeled Number of Points, enter the number of points you want to have in the zone.
3. In the text fields labeled XMin and XMax, enter the range of X-values for the defined points.
4. Click Create. A dialog informs you that the zone was created. Tecplot uniformly distributes the points along the X-axis between XMin and XMax. Y, and any other variables, are set to zero.

### 25.6.2. Creating a Rectangular Zone

Creating a rectangular zone is often the first step in interpolating irregular data into an ordered grid (see Section 21.4, “Interpolating 3-D Volume Irregular Data,” ) or in plotting analytic functions as described in Section 21.8.2, “Analytic Iso-Surface Plots.”

Tecplot allows you to create a new ordered rectangular zone with the dimensions in the I-, J- and K-directions you specify. This is done either with the Create Rectangular Zone tool or the Create Rectangular Zone dialog. The zone that you create has the same number of variables as other zones in the data set, if any. Otherwise, it has just two or three variables, depending on the value specified for the K-dimension. In the 3D frame mode, you can specify three dimensions when creating a zone. In the 2D frame mode, you can specify two dimensions. In the Sketch and XY frame modes, Create Rectangular Zone is not available, except in Sketch mode when no data set is attached to the current frame.

If you have no current data set, Tecplot creates one with two or three variables, depending on the specified K-dimension. If you specify $K=1$ (the default), the data set is created as IJ-ordered, and has just two variables. If you specify $K>1$, the data set is created as IJK-ordered, and has three variables.

**25.6.2.1. New Rectangular Zones in 2D Frame Mode.** In 2D frame mode, you may create IJ-ordered zones. You may interactively draw the boundary of the new zone, or use the Create Rectangular Zone dialog to enter the X- and Y-coordinates of the lower-left and upper-right corners of the new zone.

To create a rectangular zone interactively:

1. From the sidebar, choose the Create Rectangular Zone tool. The tool is only available if a data set is attached to the current frame. Use the Create Rectangular Zone dialog to create a zone in an empty frame.
2. In the frame in which you want to create the zone, press down and hold the left mouse button to specify one corner of the zone.
3. Drag until the boundary of the zone is as desired, then release. The Create Rectangular Zone dialog appears (see Figure 25-7) with the X and Y extents for the rectangular zone already filled in. You can adjust the I- and J-dimensions and the X- and Y-coordinates.
4. Click Create to create the zone.

To create a rectangular zone using the Create Rectangular Zone dialog:

1. From the Data menu, choose Create Zone, then Rectangular. The Create Rectangular Zone dialog appears, as in Figure 25-7.
2. In the Create Rectangular Zone dialog, you enter the number of data points in the I-direction and the number of data points in the J-direction. Also enter the X- and Y-coordinates of the two corners.
3. Click Create to create the zone.

Tecplot uniformly distributes the data points in the I- and J-directions. In this situation, the I-direction is along the X-axis, and the J-direction is along the Y-axis as shown in Figure 25-8. All other variables (those not assigned to the X- and Y-axes) are set to zero.

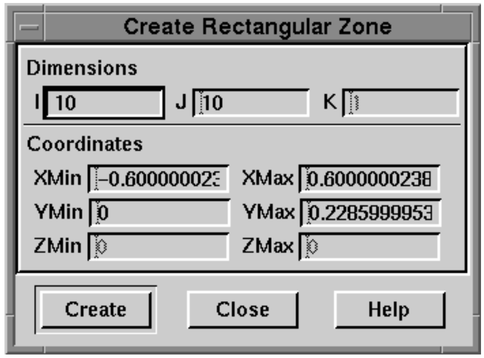



**Figure 25-7.** The Create Rectangular Zone dialog.

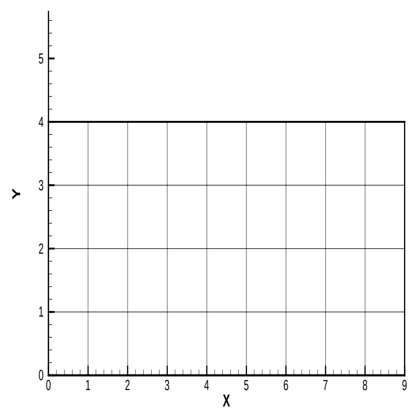



**Figure 25-8.** A rectangular zone.

After creating a rectangular zone, you can modify the X- and Y-coordinates and/or the other field variables using the tools under the Alter sub-menu of the Data menu.

**25.6.2.2. New Rectangular Zones in 3D Frame Mode.** In 3D frame mode, you can create an I-ordered (linear) zone, an IJ-ordered (planar) zone or an IJK-ordered (3-D volume) zone. In 3D frame mode, the Create Rectangular Zone mouse mode is disabled; you do not have the option to interactively draw the new zone.

To create a rectangular zone in 3D frame mode:

1. From the Data menu, choose Create Zone, then choose Rectangular. The Create Rectangular Zone dialog appears.
2. Enter the dimensions (that is, number of data points) in the I-direction, the J-direction, and the K-direction in the text fields grouped under the label Dimensions.

   To create an I-ordered zone, enter one for both the J- and K-dimensions.

   To create an IJ-ordered zone, enter one for the K-dimension. *Z=ZMin* throughout the created zone.

   To create an IJK-ordered zone, enter a K-dimension greater than one.
3. Enter the minimum and maximum coordinate values for X, Y, and Z in the text fields grouped under the label Coordinates.
4. Click Create to create the new zone.

The values for X, Y, and Z are calculated at each data point in the new zone. Tecplot distributes the data points uniformly in the I-, J-, and K-directions. In this situation, the I-direction is along the X-axis, the J-direction along the Y-axis, and the K-direction along the Z-axis. All other variables are set to zero. If you plot the new zone, you will see a rectangular mesh with uniform spacing in the X-, Y-, and Z-directions. The mesh lines are straight and parallel to the axes.

Using Alter option under the Data menu, you can modify the X-, Y-, and Z-coordinates, and the values of the other variables as well, by using equations or Equation files. See Section 25.1, “Altering Data with Equations.”

### 25.6.3. Creating a Circular or Cylindrical Zone

Tecplot allows you to create a new ordered circular or cylindrical zone with the dimensions in the I-, J-, and K-directions you specify. The I-dimension determines the number of points on each radius of the zones. The J-dimension determines the number of points around the circumference. The K-dimension determines the number of layers in the zone, creating a cylinder. You create a circular or cylindrical zone with the Create Circular Zone dialog, or, in 2D frame mode, with the Create Circular Zone tool. The zone that you create has the same number of variables as other zones in the data set, if any. In the 3D frame mode, you can specify three dimensions when creating a data set. In the 2D frame mode, you can specify two dimensions. In the Sketch and XY frame modes, the create circular zone options are not available, except in Sketch mode when no data set is attached to the current frame.

If you have no current data set, Tecplot creates one with two or three variables, depending on the K-dimension. If you specify *K=1* (the default), the data set is created as IJ-ordered, and has just two variables. If you specify *K>1*, the data set is created as IJK-ordered, and has three variables.

**25.6.3.1. New Circular Zones in 2D.** In 2D frame mode, you may create circular IJ-ordered zones. You may interactively draw the boundary of the new zone, or use the Create Circular Zone dialog to enter the radius and X- and Y-coordinates of the center of the new zone.

To create a circular zone interactively:

1. From the sidebar, choose the Create Circular Zone tool. The tool is only available if a data set is attached to the current frame. Use the Create Circular Zone dialog to create a zone in an empty frame.
2. In the frame in which you want to create the zone, press down and hold the left mouse button to specify the center of the zone.
3. Drag the mouse until the boundary of the zone is as desired, then release. The Create Circular Zone dialog appears (see Figure 25-9) with the current origin and radius filled in for you. You may adjust the dimensions and coordinates as necessary.
4. Click Create to create the new zone.

To create a circular zone using the Create Circular Zone dialog:

1. From the Data menu, choose Create Zone, then Circular. The Create Circular Zone dialog appears, as in Figure 25-9.

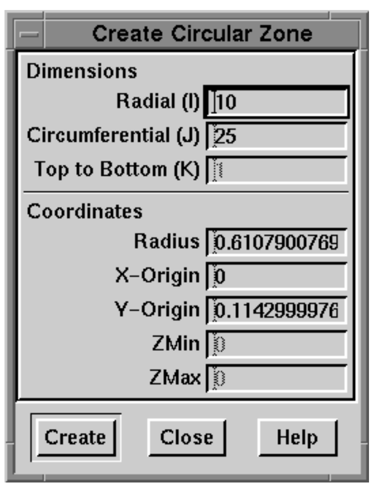



**Figure 25-9.** The Create Circular Zone dialog.

2. In the Create Circular Zone dialog, you enter the number of data points in the I-direction (radial) and the number of data points in the J-direction (circumferential). You must supply a J-dimension of at least 2.
3. Click Create to create the new zone.

Tecplot creates a zone in which I-circles are connected by J-radial lines, as shown in Figure 25-10. All other variables (those not assigned to the X- and Y-axes) are set to zero.

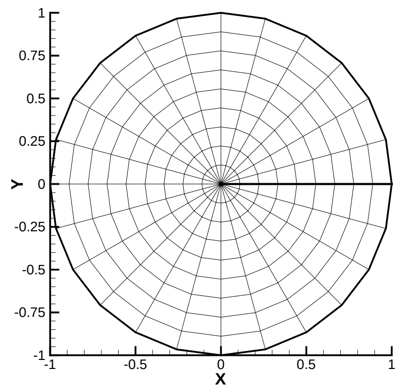



**Figure 25-10.** A circular zone.

After creating a circular zone, you can modify the X- and Y-coordinates and/or the other field variables using the tools under the Data/Alter sub-menu.

**25.6.3.2. New Cylindrical Zones in 3D Frame Mode.** In 3D frame mode, you can create an IJ-ordered (planar) zone or an IJK-ordered (3-D volume) zone. In 3D frame mode, the Create Circular Zone mouse mode is disabled, so that you do not have the option to interactively draw the new zone.

To create a cylindrical zone in 3D frame mode:

1. From the Data menu, choose Create Zone, then choose Circular. The Create Circular Zone dialog appears.
2. Enter the dimensions (that is, number of data points) in the I-direction (radial), the J-direction (circumferential), and the K-direction (layers) in the text fields grouped under the label Dimensions.

To create an IJ-ordered zone, enter one for the K-dimension. *Z=ZMin* throughout the created zone.

To create an IJK-ordered zone, enter a K-dimension greater than one.

3. Enter the radius, the X- and Y-coordinate values for the zone center, and the minimum and maximum Z coordinates in the text fields grouped under the label Coordinates.
4. Click Create to create the new zone.

The values for X, Y, and Z are calculated at each data point in the new zone. If *K>1*, Tecplot creates a K-layered cylindrical zone having I circles connected by J radial planes as shown in Figure 25-11. All other variables are set to zero.

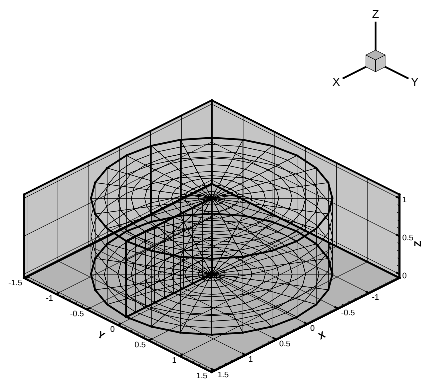



**Figure 25-11.** A 3-D circular zone.

Using the Alter option from the Data menu, you can modify the X-, Y-, and Z-coordinates, and the values of the other variables as well, by using equations or equation files. See Section 25.1, “Altering Data with Equations.”

## 25.6.4. Entering XY-Data

If you have a fairly small number of XY-pairs, you can enter them directly into Tecplot to create a zone with XY-values.

To create an I-ordered zone for XY-plots:

1. From the Data menu, choose Create Zone, then choose Enter XY. This action calls up the Enter XY-Values to Create a Zone dialog, as shown in Figure 25-12.

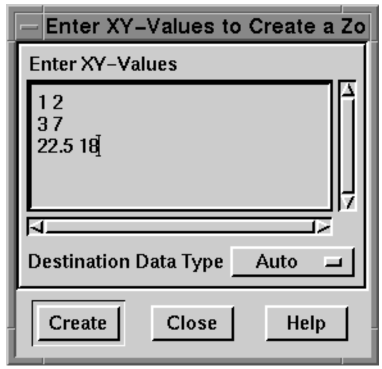



**Figure 25-12.** The Enter XY-Values to Create a Zone dialog.

2. In the text box labeled Enter XY Values, enter X- and Y-value pairs, one per line; first X, then one or more spaces, then Y.
3. If you would like to specify a data type for the data (long or short integer, float, double, byte, bit), select the desired data type from the drop-down labeled Destination Data Type.
4. Click Create to create the zone.

### 25.6.5. Extracting Data Points

Another method for creating an I-ordered zone is to extract data points from the current data set using any of three methods:

- Choosing a discrete set of points with the mouse.
- Drawing a polyline with the mouse from which points are extracted at regular intervals.
- Selecting an existing polyline geometry from which points are extracted, either at regular intervals or at the points which define the geometry.

The difference between the second and third methods is that in the second method, the polyline used to extract the points is not a geometry and is not part of your plot—it is simply a transient structure used to extract the points. The points which define the polyline are not treated specially in any way. When you extract points from a polyline geometry, however, you can choose to use the points that define the polyline as the extracted points.

**Note:** To extract points from a geometry or polyline, it must lie within the boundaries of a zone with connectivity.

**25.6.5.1. Extracting Discrete Points.** To extract a discrete set of points with the mouse:

1. From the Data menu, choose Extract, then choose Discrete Points.
2. Click at each location from which you want to extract a point.
3. Double-click on the last data point or press Esc to end. The Extract Data Points dialog appears; use it to specify how to save the data.

**25.6.5.2. Extracting Points from Polyline.** To extract points from a polyline:

1. From the Data menu, choose Extract, then choose Points from Polyline.
2. Click at the desired beginning of the line, and at all desired breakpoints.
3. Double-click on the last data point or press Esc to end. The Extract Data Points dialog appears; use it to specify how many points to extract and how to save the data.

**25.6.5.3. Extracting Points from Geometry.** To extract points from a polyline geometry:

1. In the workspace, select the polyline geometry from which you want to extract data points.
2. From the Data menu, choose Extract, then choose Points from Geometry. The Extract Data Points dialog appears; use it to specify how many points to extract and how to save the data.

**25.6.5.4. Controlling Data Point Extraction.** Use the Extract Data Points dialog to control how data points are extracted. Use the following controls:

- **Extract Data to:**
  - **File:** Select this check box if you want the data points extracted to an ASCII Tecplot data file. If this check box is selected, the Extract Data Points to File dialog appears when you click Extract. Use this dialog to specify a file name for the extracted data file.
  - **Zone:** Select this check box if you want the data points extracted to a zone in the current data set.
- **Include distance variable:** Select this check box if you want the extracted data file to contain an additional variable, **`DISTANCE`**, which contains the accumulated distance from the first point.
- **Number of points to extract:** Enter the number of points to extract. This field is sensitive only if you are extracting data points from a polyline or geometry. It is insensitive if you are extracting discrete points. If you are extracting from a geometry, you must also select the check box labeled “Extract regular points along a geometry.”

- **Extract regular points along geometry:** Select this check box if you want to extract the specified number of points distributed uniformly along the geometry. This check box is sensitive only if you are extracting points from a geometry.
- **Extract only points which define geometry:** Select this check box if you want to extract only the endpoints of the segments in the geometry. This check box is sensitive only if you are extracting points from a geometry.

After specifying any desired options, click Extract.

### 25.6.6. Duplicating a Zone

Tecplot can create a new zone by duplicating all or part of an existing zone. This is useful for creating projections, mirror images, and subzones of existing zones.

**25.6.6.1. Duplicating a Full Zone.** To create a full duplicate of one or more existing zones:

1. From the Data menu, choose Create Zone, then Duplicate. The Create Duplicate Zone dialog appears, as shown in Figure 25-13.

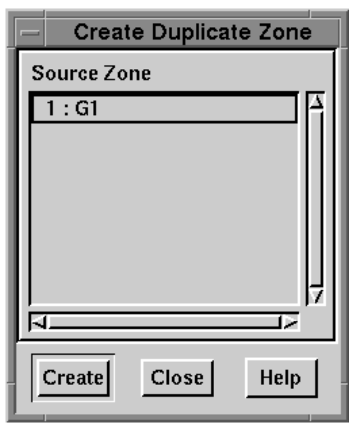



**Figure 25-13.** The Create Duplicate Zone dialog.

2. Select the source zone or zones from the scrolled list labeled Source Zone.
3. Click Create to create the duplicate zone or zones. Each duplicate zone has the same name as the zone of which it is a copy.

Duplicate zones can be used as the first step to create new zones which are projections of existing 3-D surface zones onto a plane. To do this, follow these steps:

1. Create a plot of a 3-D surface zone.
2. Create a duplicate of the surface zone using the procedures in Section 25.6.6.1, “Duplicating a Full Zone.”
3. From the Data menu, choose Alter, then choose Specify Equations.
4. Make the new zone a plane by setting the Z-coordinate to a constant value (-20,000 in this example) by entering the following equation in the Specify Equations dialog:

```
Z = -20000
```

5. Select the new zone in the Zones to Alter region of the Specify Equations dialog, deselecting other zones as necessary.
6. Click Compute to complete modifying the new zone.

Figure 25-14 shows an example of a projection of a 3-D surface zone.

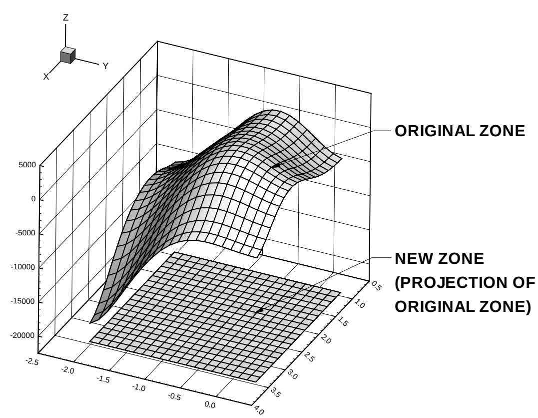



**Figure 25-14.** Projection of a 3-D surface.

**25.6.6.2. Creating a Sub-zone.** You can only create a sub-zone out of an existing I-, IJ-, or IJK-ordered zone. You cannot create a sub-zone out of a finite-element zone.

To create a sub-zone of an existing zone:

1. From the Data menu, choose Create Zone, then SubZone. The Create SubZone dialog appears, as shown in Figure 25-15.
2. Select the source zone from the Source Zone drop-down menu.

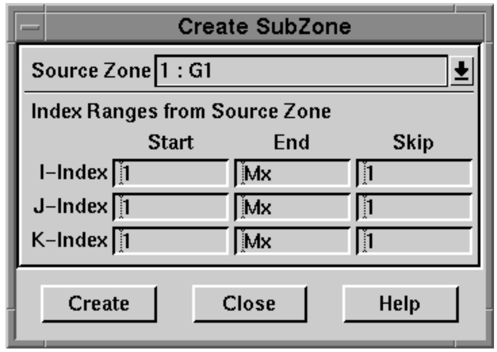



**Figure 25-15.** The Create SubZone dialog.

3. Specify the desired sub-zone as a range of I-, J-, and K-indices. For each of I-Index, J-Index, and K-Index (if applicable), specify a start index, an end index, and a skip. You may use the special value **`0`** or **`Mx`** to indicate the maximum of that index, and the values **`Mx-1`** to represent one index less than the maximum, **`Mx-2`** for two less than the maximum, and so forth.
4. Click Create to create the subzones. Each sub-zone is given the name “SubZone.”

**25.6.6.3. Creating a Mirror Zone.** Tecplot makes it very simple to create a duplicate zone that is the mirror image of an existing zone if the desired mirror axis or mirror plane is one of the standard axes (2-D) or the plane determined by any two axes (3-D). To create a mirror image of one or more existing zones using one of these standard mirrors:

1. From the Data menu, choose Create Zone, then Mirror. The Create Mirror Zone dialog appears, as shown in Figure 25-16.
2. Select the source zone or zones from the scrolled list labeled Source Zone(s).
3. Specify the axis (2-D) or axis plane (3-D) to mirror about.
4. Click Create to create the mirror zone or zones. Each mirror zone has a name of the form “Mirror of zone sourcezone,” where sourcezone is the number of the zone from which the mirrored zone was created.

For example, consider the case where your input file describes a 90 degree wedge which occupies the positive X-positive Y-quadrant. You want to create the complete circle based on your inputs. To do this, perform the following steps:

1. Mirror the original zone about the X-axis.
2. Mirror both the original zone and the mirrored zone created in Step 1 about the Y-axis.

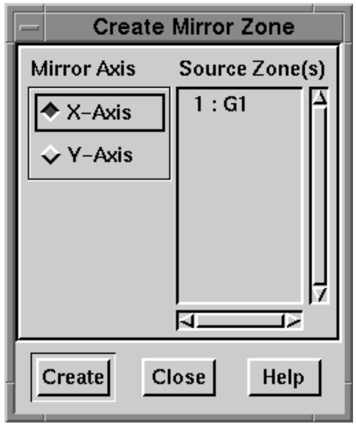



**Figure 25-16.** The Create Mirror Zone dialog for a 2-D image.

It is also possible to create a mirror zone that is mirrored about a different axis or plane. For example, suppose you wanted to mirror a 2-D zone about the line **`X=5`**. You can do this as follows:

1. Create a mirror zone about the Y-axis as described above
2. From the Data menu, choose Alter, then choose Specify Equations.
3. Enter the following equation in the Equations text field:

   ```
   X = 10 - X
   ```

4. Specify the new mirror zone as the only zone to alter.
5. Click Compute to complete the process.

Figure 25-17 shows an example of creating mirror image zones.

## 25.7. Deleting Zones

In any data set with more than one zone, you can use the Delete Zone dialog to delete any unwanted zones. You cannot delete all zones; if you attempt to delete all zones, the lowest numbered zone is not deleted.

To delete a zone:

1. From the Data menu, choose Delete Zone. The Delete Zone dialog appears as shown in Figure 25-18.

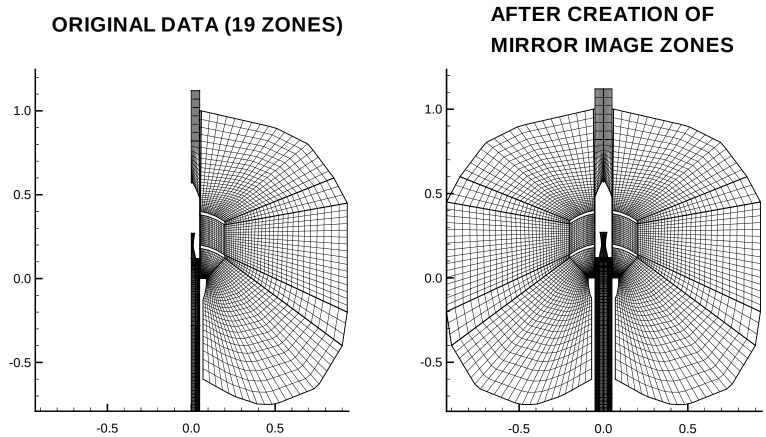



**Figure 25-17.** Creating a mirrored zone.

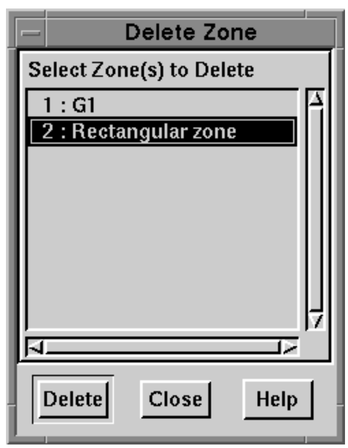



**Figure 25-18.** The Delete Zone dialog.

2. Select the zone or zones you want to delete.
3. Click Delete to delete the zones.

## 25.8. Triangulating Irregular Data Points

Triangulation is a process in which data points are connected to form triangles. You can use triangulation to convert irregular, I-ordered data sets into a finite-element surface zone. You can, in fact, use triangulation on any type of source zone, ordered or finite-element. But its use on irregular data is most common. Triangulation is one of two options for creating 2-D field plots from irregular data. The other is interpolation, discussed in Section 25.9, “Interpolating Data.” Triangulation preserves the accuracy of the data by creating an finite-element surface zone with the source data points as nodes and a set of Triangle elements. Triangulation is only available in the 2D frame mode.

Triangulation works best for 2-D data; you can, however, triangulate 3-D surface data as long as the Z-coordinate is single-valued (the surface does not wrap around on itself). When you triangulate 3-D surface data, the Z-coordinate of the data is ignored, causing a less-than-optimal triangulation in some cases.

To triangulate your data:

1. From the Data menu, choose Triangulate. The Triangulate dialog appears, as shown in Figure 25-19.

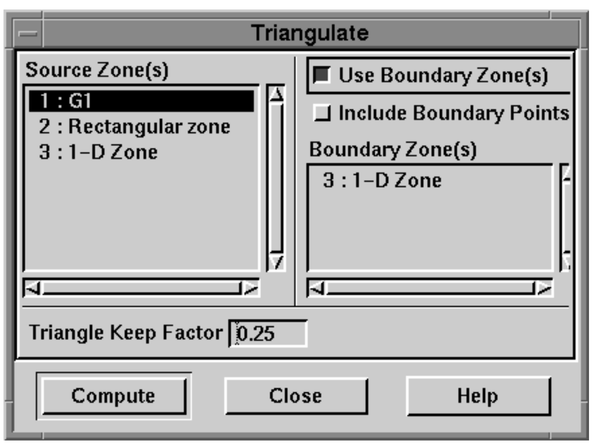



**Figure 25-19.** The Triangulate dialog.

2. Select the zone or zones to triangulate from the scrolled list labeled Source Zone(s).

3. If you want to specify a boundary zone for the triangulation, select the Use Boundary Zone(s) check box and select the boundary zone or zones from the Boundary Zone(s) scrolled list. The boundary zones define the boundaries in the triangulation region; if you do not include boundary zones, Tecplot assumes the data points lie within a convex polygon and that all points in the interior can be connected.
4. (If Use Boundary Zone(s) is selected) If you want to include the points in the boundary zones in the triangulated zone, select the Include Boundary Points check box.
5. (Optional) Modify the Triangle Keep Factor, if desired. This factor is used to define "bad" triangles on the outside of the triangulated zone. See below for complete details.
6. Click Compute to perform the triangulation.

At the completion of triangulation, Tecplot attempts to remove bad triangles from the outside of the triangulation. This does not include any triangles next to the boundary zone, only those along the edges where there is no boundary (or where the boundary zone points are excluded). The definition of a bad triangle is stored as the Triangle Keep Factor as a number between zero (three collinear points) and 1.0 (an equilateral triangle). Typical settings are values between 0.1 and 0.3; settings above 0.5 are not allowed.

After triangulating your data, you can use the resulting finite-element surface zone to create plots. Generally, you turn off the original zone(s) and plot the new zone only, but you can, for example, plot a scatter plot of the original zone(s) along with the contours of the new zone.

## 25.9. Interpolating Data

Interpolation, in Tecplot, means assigning new values for the variables at data points in a zone based on the data point values in another zone (or set of zones).

For example, you may have a set of data points in an I-ordered zone that are distributed in a random-like fashion in the XY-plane. This type of data is sometimes referred to as unordered, ungridded, or random data; in Tecplot, it is called irregular data. Using data in this form, you can create mesh plots and scatter plots, but you cannot create contour plots, light-source shading, or streamtraces. In Tecplot, you can interpolate the irregular I-ordered data onto an IJ-ordered mesh, and then create contour plots and other types of field plots with the interpolated data. You can also interpolate your 3-D, I-ordered irregular data into an IJK-ordered zone and create 3-D volume plots from the IJK-ordered zone. You can even interpolate to a finite-element zone.

There are three types of interpolation available:

- **Linear:** Interpolate using linear interpolation from a set of finite-element, IJ-ordered, or IJK-ordered zones to one zone.

- **Inverse Distance:** Interpolate using an inverse-distance weighting from a set of zones to one zone.
- **Kriging:** Interpolate using kriging from a set of zones to one zone.

Each of these options is described in the following sections.

### 25.9.1. Inverse-Distance Interpolation

Inverse-distance interpolation averages the values at the data points from one set of zones (the source zones) to the data points in another zone (the destination zone). The average is weighted by a function of the distance between each source data point to the destination data point. The closer a source data point is to the destination data point, the greater its value is weighted.

In many cases, the source zone is an irregular data set—an I-ordered set of data points without any mesh structure (a list of points). Inverse-distance interpolation may be used to create 2- or 3-D surface, or a 3-D volume field plots of irregular data. The destination zone can, for example, be a circular or rectangular zone created within Tecplot. See Section 25.6, “Creating Zones.”

To perform inverse-distance interpolation in Tecplot, use the following steps:

1. Read the data set to be interpolated into Tecplot (the source data).
2. Read in or create the zone onto which the data is to be interpolated (the destination zone).
3. From the Data menu, choose Interpolate, then choose Inverse Distance. The Inverse-Distance Interpolation dialog appears, as shown in Figure 25-20.
4. Select the zones to be interpolated from those listed in the Source Zone(s) scrolled list.
5. Select which variables are to be interpolated from those listed in the Variable(s) scrolled list. By default, all variables are interpolated except those assigned to the X-, Y-, and Z-axes. If, after interpolating, you will be working with just one or two interpolated variables, you can speed up the calculations by interpolating only those needed variables.
6. Select the destination zone into which to interpolate. Existing values for the interpolated variables in the destination zone will be overwritten.
7. (Optional) Enter the minimum distance used for the inverse-distance weighting in the Minimum Distance text field. Source data points which are closer to a destination data point than this minimum distance are weighted as if they were at the minimum distance, thus reducing the weighting factor for such points. This tends to reduce the peaking and plateauing of the interpolated data near the source data points.
8. (Optional) Enter the exponent for the inverse-distance weighting in the Exponent text field.
9. (Optional) Select the method used for determining which source points to consider for each destination point from the Point Selection drop-down menu. There are three available methods, as follows:

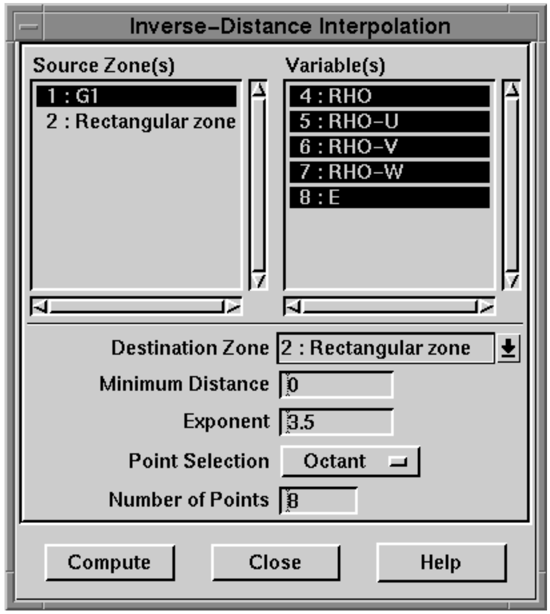



**Figure 25-20.** The Inverse-Distance Interpolation dialog.

- **Nearest N:** For each point in the destination zone, consider only the closest *n* points to the destination point. These *n* points can come from any of the source zones. You specify *n* after selecting this option. This option may speed up processing if *n* is significantly smaller than the entire number of source points.
- **Octant:** Like Nearest N above, except the *n* points are selected by coordinate-system octants. The *n* points are selected so they are distributed as evenly as possible throughout the eight octants. This reduces the chances of using source points which are all on one side of the destination point.
- **All:** Consider all points in the source zone(s) for each point in the destination zone.

10. (Optional) If you specified Nearest N or Octant for the point selection method, enter the number of points.
11. Click Compute to perform the interpolation. While the interpolation is proceeding, a working dialog appears showing the progress of the interpolation. This dialog has a Cancel button allowing you to interrupt the interpolation.

    If you click Cancel during the interpolation process, the interpolation is terminated prematurely. The destination zone is left in an indeterminate state, and you should redo the interpolation.

Inverse-distance interpolation ignores the IJK-mode of IJK-ordered zones. All data points in both the source and destination zones are used in the interpolation.

**Note:** Tecplot uses the current frame's axis assignments to determine the variables to use for coordinates in interpolation, but it ignores any axis scaling that may be in effect.

**25.9.1.1. The Inverse-Distance Algorithm.** The algorithm used for inverse-distance interpolation is simple. The value of a variable at a data point in the destination zone is calculated as a function of the selected data points in the source zone (as defined in the Point Selection drop-down).

The value at each source zone data point is weighted by the inverse of the distance between the source data point and the destination data point raised to a power as shown below:

$$\varphi_d = \frac{\sum w_s \varphi_s}{\sum w_s} \quad \text{(summed over the selected points in the source zone)}$$

where $\varphi_d$ and $\varphi_s$ are the values of the variables at the destination point and the source point, respectively, and $w_s$ is the weighting function defined as:

$$w_s = D^{-E}$$

$D$ in the equation above is the distance between the source point and the destination point or the minimum distance specified in the dialog, whichever is greater. $E$ is the exponent specified in the Exponent text field. The exponent should be set between 2 and 5. The algorithm is speed-optimized for an exponent of 4, although in many cases, the interpolation looks better with an exponent of 3.5.

Smoothing may improve the data created by inverse-distance interpolation. Smoothing adjusts the values at data points toward the average of the values at neighboring data points, removing peaks, plateaus, and noise from the data. See Section 25.10, "Smoothing Data," for information on smoothing.

### 25.9.2. Kriging

Kriging is a more complex form of interpolation than inverse-distance. It works similar to inverse-distance interpolation discussed in Section 25.9.1, "Inverse-Distance Interpolation," and is used for the same purposes. Kriging generally produces superior results to the inverse-distance algorithm but requires more computer memory and time.

To perform kriging in Tecplot, perform the following steps:

1. Read the data set to be interpolated into Tecplot (the source data).
2. Read in or create the zone onto which the data is to be interpolated (the destination zone).
3. From the Data menu, choose Interpolate, then choose Kriging. The Kriging dialog appears, as shown in Figure 25-21

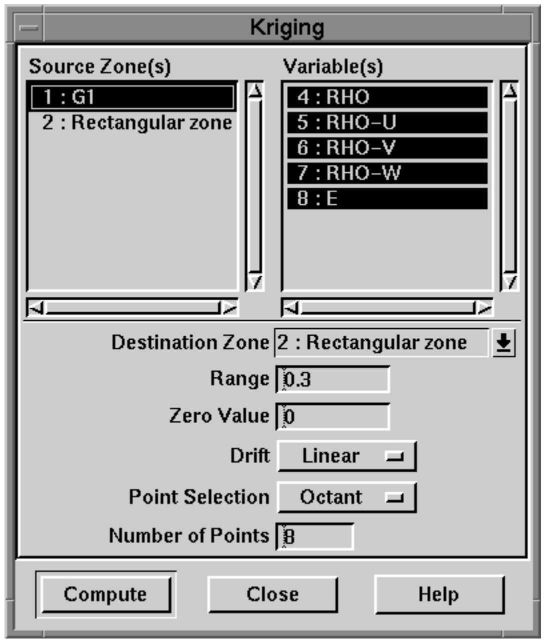



**Figure 25-21.** The Kriging dialog in Motif.

4. Select the zones to be interpolated from those listed in the Source Zone(s) scrolled list.
5. Select which variables are to be interpolated from those listed in the Variable(s) scrolled list. By default, all variables are interpolated except those assigned to the X-, Y-, and Z-axes. If, after interpolating, you will be working with just one or two interpolated variables, you can speed up the calculations by interpolating only those needed variables. In kriging, interpolating fewer variables can have a significant effect on the speed of interpolation.
6. Select the destination zone into which to interpolate. Existing values for the interpolated variables in the destination zone will be overwritten.
7. (Optional) In the Range text field, enter the distance beyond which source points become insignificant for the kriging. The value is stated as the fraction of the length of the diagonal of the box which contains the data points. A range of zero means that any point not coinci-

dent with the destination point is statistically insignificant; a range of one means that every point in the data set is statistically significant for each point. In general, values between 0.2 and 0.5 should be used.

8. (Optional) In the Zero Value text field, enter the semi-variance at each source data point on a normalized scale from zero to one. Semi-variance is the certainty of the value at a data point. A value of zero means that the values at the source points are exact. Greater values mean the values at the source points have some uncertainty or noise. Zero is usually a good number for the zero value, and it causes the interpolated data to fit closely to all the source data points. Increasing the zero value results in smoother interpolated values that fit increasingly more to the average of the source data.
9. (Optional) Select the overall trend for the data in the Drift drop-down. This can be No Drift, Linear, or Quadratic.
10. (Optional) Select the method used for determining which source points to consider for each destination point from the Point Selection drop-down. There are three available methods, as follows:
    - **Nearest N:** For each point in the destination zone, consider only the closest *n* points to the destination point. These *n* points can come from any of the source zones.
    - **Octant:** Like Nearest N above, except the *n* points are selected by coordinate-system octants. The *n* points are selected so they are distributed as evenly as possible throughout the eight octants. This reduces the chances of using source points which are all on one side of the destination point.
    - **All:** Consider all points in the source zone(s) for each point in the destination zone.

    This option is very important for kriging, since kriging involves the computationally expensive inversion and multiplication of matrices. The computational time and memory requirements increase rapidly as the number of selected source data points increases. In general, you should not use the All option unless you have very few source points.
11. If you specified Nearest N or Octant for the point selection method, enter the number of points.
12. Click Compute to perform the kriging. While the kriging is proceeding, a working dialog appears showing its progress. This dialog has a Cancel button allowing you to interrupt the kriging.

    If you click Cancel during the kriging process, the kriging is terminated prematurely. The destination zone is left in an indeterminate state, and you should redo the kriging.

**Note:** Tecplot uses the current frame’s axis assignments to determine the variables to use for coordinates in kriging, but it ignores any axis scaling that may be in effect. Also, if the Drift is set to Linear or Quadratic, Tecplot requires that the points selected be non-collinear (non-coplanar in 3-D). To avoid this limitation, set the Drift to None. Tecplot requires that no points be coincident. You can eliminate coincident points by triangulation before you interpolate.

**25.9.2.1. The Kriging Algorithm.** For a detailed discussion of the kriging algorithm see:

> Davis, J. C., *Statistics and Data Analysis in Geology*, Second Edition, John Wiley & Sons, New York, 1973, 1986.

**25.9.2.2. Improving Results with Kriging and Inverse Distance Interpolation.** For better results with 3-D data, try changing the range of your Z-variable to one similar to the X-range the Y-range. Also, set Zero Value to 0.05.

## 25.9.3. Linear Interpolation

Linear interpolation differs from the two previous interpolation schemes previously discussed in that the source zone must have some 2- or 3-D structure. That is, the source zones must be IJ-ordered, IJK-ordered, or finite-element. Irregular I-ordered data cannot be used for the source zones. (For 2-D data, you may be able to first create a finite-element zone from an irregular, I-ordered zone by using triangulation. See Section 25.8, “Triangulating Irregular Data Points.”)

Linear interpolation finds the values in the destination zone based on their location within the cells of the source zones. The value is linearly interpolated to the destination data points using only the data points at the vertices of the cell (or element) in the source zone(s).

To perform linear interpolation in Tecplot, perform the following steps:

1. Read the data set to be interpolated into Tecplot (the source data).
2. Read in or create the zone onto which the data is to be interpolated (the destination zone).
3. From the Data menu, choose Interpolate, then choose Linear. The Linear Interpolation dialog appears, as shown in Figure 25-22.
4. Select the zones to be interpolated from those listed in the Source Zone(s) scrolled list.
5. Select which variables are to be interpolated from those listed in the Variable(s) scrolled list. By default, all variables are interpolated except those assigned to the X-, Y-, and Z-axes. If, after interpolating, you will be working with just one or two interpolated variables, you can speed up the calculations by interpolating only those needed variables.
6. Select the destination zone into which to interpolate. Existing values for the interpolated variables in the destination zone will be overwritten.
7. (Optional) Select what to do with points that lie outside the source-zone data field. You have two options, represented by option buttons on the dialog: Constant, which sets all points outside the data field to a constant value that you specify; and Do Not Change, which preserves the values of points outside the data field. Do Not Change is appropriate in cases where you are using one interpolation algorithm inside the data field, and another outside.
8. (Optional) If you choose Constant as the Outside Points option, specify the constant value in the Constant Value text field.

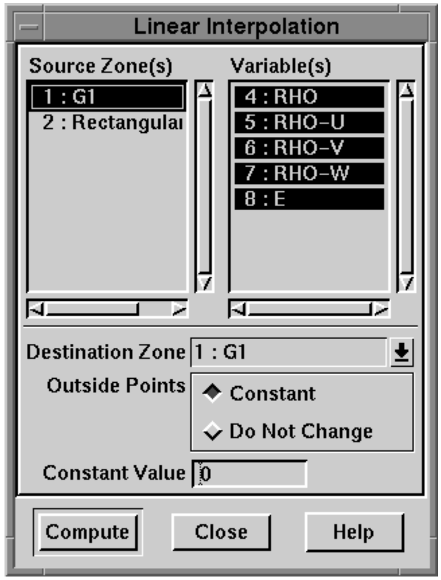



**Figure 25-22.** The Linear Interpolation dialog.

9. Click Compute to perform the interpolation. While the interpolation is proceeding, a working dialog appears showing the progress of the interpolation. This dialog has a Cancel button allowing you to interrupt the interpolation.

   If you click Cancel during the interpolation process, the interpolation is terminated prematurely. The destination zone is left in an indeterminate state, and you should redo the interpolation.

### 25.9.4. Alternatives to Interpolation

An alternative to 2-D interpolation is to triangulate your irregular data points. This creates a mesh of triangles (a Triangle element-type, finite-element zone) using the source data as node points. No interpolation is required. See Section 25.8, "Triangulating Irregular Data Points," for a description of triangulation.

## 25.10. Smoothing Data

You can smooth the values of a variable of an I-, IJ-, or IJK-ordered zone (in either 2- or 3-D) to reduce "noise" and lessen discontinuities in data. Smoothing can also be used after inverse-distance interpolation to reduce the artificial peaks and plateaus.

Smoothing is applied to XY-lines, in 2-D, across a 3-D surface, or in a 3-D volume, depending on the state of the current frame and the type of zone structure. Each pass of smoothing shifts the value of a variable at a data point towards an average of the values at its neighboring data points.

To smooth data in Tecplot, use the following steps:

1. From the Data menu, choose Alter, then choose Smooth. The Smooth dialog appears as shown in Figure 25-23.

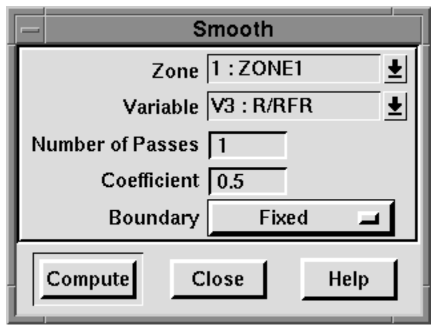



**Figure 25-23.** The Smooth dialog.

2. Select the zone to smooth from the Zone drop-down. The zone must be I-, IJ-, or IJK-ordered. The zone should not intersect itself.
3. Select the variable to smooth from the drop-down of variables in the zone. In XY frame mode, the variable must be a dependent variable for one active mapping for that zone.
4. (Optional) Specify the number of smoothing passes to perform. The default is 1. A greater number of passes will take more time but smooth the data more.
5. (Optional) Specify the relaxation factor for each pass of smoothing in the field labeled Coefficient. Enter a number between zero and one (exclusively). Large numbers flatten peaks and noise quickly. Small numbers smooth less each pass, rounding out peaks and valleys rather than eliminating them.
6. (Optional) Select the boundary conditions by which to smooth from the Boundary drop-down. The options are Fixed, First Order, and Second Order, as follows:
   - **Fixed:** The points at the boundary are not changed in value.
   - **First Order:** The points at the boundary are smoothed based on the assumption that the first derivative normal to the boundary is constant. This will tend to cause contour lines of the smoothed variable to be perpendicular to the boundary.

- **Second Order:** The points at the boundary are smoothed based on the assumption that the second derivative normal to the boundary is constant. This option may over-extrapolate derivatives at the boundary.

7. Click Compute to perform the smoothing. While the smoothing is underway, a working dialog appears showing the progress of the smoothing. This dialog has a Cancel button allowing you to interrupt the smoothing.

   If you click Cancel during the smoothing process, you will interrupt the smoothing, and Tecplot will report back the number of passes completed. For example, if you specified ten passes in Number of Passes but hit escape halfway through, Tecplot would report five passes complete.

Smoothing has some limitations:

- Finite-element zones cannot be smoothed. Only I-, IJ-, and IJK-ordered zones can be smoothed.
- Tecplot uses the current frame's axis assignments to determine the variables to use for the coordinates in the smoothing, and also to determine whether the smoothing should be done in XY, 2D, or 3D frame mode. Be careful if you have multiple frames with different variable assignments for the same data set.
- Any axis scaling is ignored by Tecplot while smoothing.
- For I-ordered zones, the current frame can be in XY, 2D, or 3D frame mode. In XY mode, the variable must be the dependent variable of one active mapping for that zone.
- For IJ-ordered zones, the current frame can be in 2D or 3D frame mode, but you cannot smooth the variables assigned to the X- and Y-axes in the 2D frame mode.
- For IJK-ordered zones, you must be in 3D frame mode, and you cannot smooth the variables assigned to the X-, Y-, and Z-axes. The IJK-mode is ignored. The zone is smoothed with respect to the entire 3-D volume.
- Smoothing does not extend across zone boundaries. If you use a boundary condition option other than Fixed (such that values along the zone boundary change), contour lines can be discontinuous at the zone boundaries.
- Smoothing is performed on all nodes of a zone, and disregards value-blanking if it is active.

# CHAPTER 26 Probing

In Tecplot probing is the ability to select a point and view the values of all variables at that point. You can also view information about the data set itself while probing. Similar to probing is probing-to-edit, a feature which allows you to modify your data interactively. To prevent you from inadvertently changing your data, probing-to-edit is disabled by default. You can enable the feature by toggling the option Allow Data Point Adjustment in the Edit menu. Use either the Probe At dialog or the Probe tool to obtain point information from a data field.

With the Probe At dialog, you can specify the location of the probe as either a set of spatial coordinates X, Y, and Z, or as a set of I-, J-, and K-indices. You select one or more locations in the data field where information is to be collected, and the resulting information is displayed in the Probe dialog.

When you probe with the mouse, you can probe in either of two modes: Interpolate and Nearest Point. In Interpolate mode, accessed by a mouse click, the value returned is the linearly interpolated value for the specified locations. In Nearest Point mode, accessed by Ctrl-click, the value returned is the exact value at the closest data point in the field.

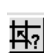
## 26.1. Probing Field Plots with the Mouse

The most direct method of probing is to use the Probe tool. When you select the Probe tool from the sidebar and move the pointer into the workspace, where it becomes a cross-hair. Click at any point to probe in Interpolate mode, which calls up a dialog showing the probe information interpolated for that point. Ctrl-click at any location to probe in Nearest Point mode which will obtain probe information for the data point closest to the cross-hair.

The following table shows the information returned for each type of probe action for field plots. (All mouse click operations are using the left mouse button.)

| Probe Action | Information Returned |
|---|---|
| Click | If the pointer is over a valid cell return the interpolated field values from all nodes in the cell. If multiple cells are candidates then, for 2D frame mode the cell from the highest number zone is used and for 3D frame mode the cell closest to the viewer is used. |
| Ctrl-Click | If the pointer is over a valid cell return the field values from the nearest node in the cell. If multiple cells are candidates then, for 2D frame mode the cell from the highest number zone is used and for 3D frame mode the cell closest to the viewer is used. If the pointer is not over any cell then the field values from nearest data point as measured in distance on the screen are returned. |
| Shift-Ctrl-Click | Return the field values from the nearest point on the screen ignoring surfaces and regardless of zone number or depth of the point. This is useful in 3-D for probing on data points that are on the back side of a closed surface without having to rotate the object. In 2-D this is useful for probing on data points for zones that may be underneath other zones because of the order in which they were drawn. |
| Alt-Click<br>(3D Frame mode only) | Same as Click except ignore zones while probing. (Probe only on streamtraces, iso-surfaces, or slices.) |
| Alt-Ctrl-Click | Same as Ctrl-Click except ignore zones while probing. (Probe only on streamtraces, iso-surfaces, or slices.) |
| Alt-Ctrl-Shift-Click | Same as Shift-Ctrl-Click except ignore zones while probing. (Probe only on streamtraces, iso-surfaces, or slices.) |

To obtain interpolated variable values for the exact probed location:

1. Select the Probe tool, represented by , from the sidebar.
2. Move the pointer into the workspace. The pointer changes to a cross-hair.
3. Click at the desired location. A cross appears at the probed location, and the Probe dialog appears as shown in Figure 26-1, showing variable values. The variable values are interpolated linearly from the values of the data set. If you probe a 3-D volume zone in 3D frame mode, the probe cross-hairs point to locations on the surface of the plot, not to locations within the plot.
4. To view zone and cell information, click Zone/Cell Info to bring up the Zone/Cell Info page, as shown in Figure 26-1.

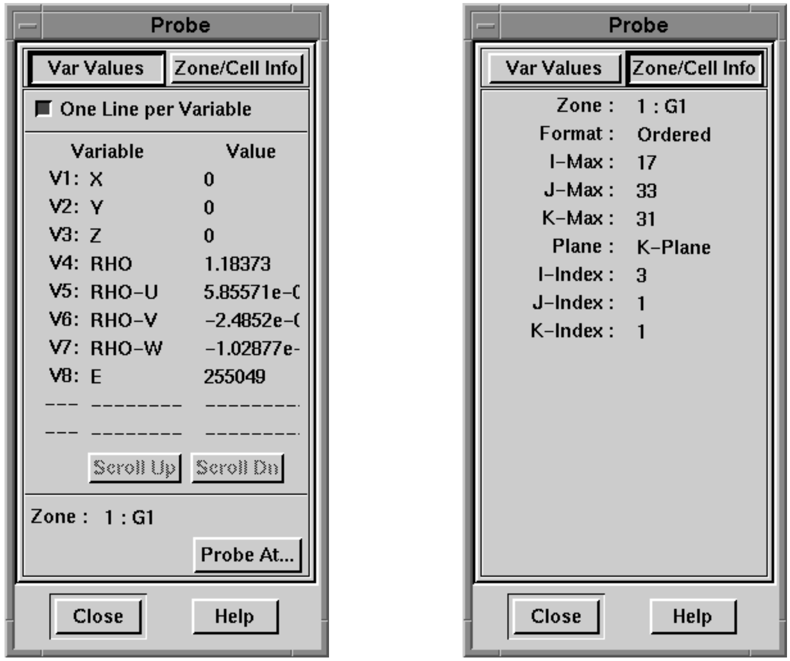
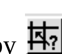



**Figure 26-1.** The Var Values and Zone/Cell Info pages of the Probe dialog.

5. Click at additional locations to view variable values or zone and cell information for other data points.

**Note:** Interpolate mode does not work for I-ordered data displayed in 2D or 3D frame mode; if you probe such data you will always get the error message “Point is outside of data field,” because Tecplot cannot interpolate without a field mesh structure. You can, however, use the Nearest Point mode (described below) in such situations.

To obtain exact results for the data point nearest the probed location:

1. On the sidebar select the Probe tool, represented by .
2. Move the pointer into the workspace where it will change to a cross-hair.
3. Ctrl-click at the desired location. An XORed cross appears at the data point closest to the probed location, and the Probe dialog appears showing variable values, including the X-, Y-, and Z-coordinates of the nearest point.

The variable values are the exact values for the nearest point. If the zone is not I-ordered and the cross-hairs are placed within the data field, the point reported is the closest data point in the cell pointed to by the pointer. (This may not be the closest point in the entire data field.) If you probe a 3-D volume zone in 3D frame mode, the probe cross-hairs point to locations on the surface of the plot, not to locations within the plot.

4. To view zone and cell information, click the Zone/Cell Info to call up the Zone/Cell Info page.
5. Ctrl-click at additional locations to view variable values or zone and cell information for other data points.

You may alternate between interpolated and exact values by clicking and Ctrl-clicking.

## 26.2. Advanced Probing

By default a Tecplot probe first detects a zone cell face. It then finds the nearest point of that face if Ctrl-click was used while probing. However, advanced probe options let you probe points behind a cell face, or objects that are contained within zones.

### 26.2.1. Probing Obscured Points

Nearest point probing using Ctrl-click is limited in two ways. In 2D frame mode the Probe tool will select mesh intersection points only according to drawing order when using Ctrl-click. By default, when probing data in 3D frame mode using Ctrl-click with the Probe tool, the probe will only select the surface nearest you. Thus, if you are viewing a 3-D wire mesh, the probe will not select mesh intersection points shown on the far side of an enclosed surface, and you would need to rotate the view in order to select points on the far side.

To overcome this, you may use Shift-Ctrl-click. This option lets you select points to probe as if all visible points were projected onto the 2-D plane of the screen and a ruler was laid on the screen to measure the distance from the cross-hair to each point, allowing you to find the nearest point.

### 26.2.2. Probing on Streamtraces, Iso-Surfaces, and Slices

In 2D frame mode Shift-Ctrl-click allows users to probe mesh intersection points regardless of drawing order. In 3D frame mode it will allow you to chose mesh intersection points independent of surface depth. Thus, when viewing a 3-D wire mesh for example, you may use Shift-Ctrl-click to select points on the far side of an enclosed surface.

You may probe for values on volume objects such as streamtraces, iso-surfaces and slices, with the Alt key. Using the Alt key alone when probing will give you interpolated values of the

nearest volume object. Using Alt-Ctrl-click will probe a point where the mesh intersects the closest object. Using Alt-Shift-Ctrl-click will probe the closest point where the mesh intersects any object.

## 26.3. Probing Field Plots by Specifying Coordinates and Indices

If you want precise control over your probe location, or if you want to probe using I-, J-, and K-indices, or if you want to probe inside a 3-D volume, you need to use the Probe At dialog to specify the probe location. You can launch the Probe At dialog either from the Data menu (by choosing Probe At), from the Var Values page of the Probe dialog (using the Probe At button), or by clicking Details in Probe mode.

To probe at a specified location using spatial coordinates (in Interpolate mode):

1. Launch the Probe At dialog. The Probe At dialog appears, as in Figure 26-2, ready for you to enter X-, Y-, and Z-coordinates.
2. Enter the X-, Y-, and Z-coordinates of the desired probe location.
3. If the zone you are probing is a 3-D volume zone, select the check box labeled Probe Within Volume to ensure that the probe is performed at the indicated point. If you specify a position within a 3-D volume zone and the Probe Within Volume check box is not selected, Tecplot probes at the surface of the zone nearest the user's eye along the ray defined by the specified point and the user's eye.

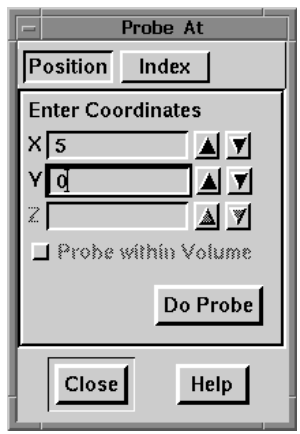



**Figure 26-2.** The Position page of the Probe At dialog.

4. Click Do Probe to perform the probe. The Probe dialog appears with interpolated values for the specified location.

To probe at a specified location using data set indices (in Nearest Point mode):

1. Launch the Probe At dialog. The Probe At dialog appears.
2. Click the Index button (Index tab in Windows) to bring up the Index page of the Probe At dialog, shown in Figure 26-3.

3. Select the desired zone from the Zone drop-down.
4. Enter the I-, J-, and K-indices of the desired probe location. (For finite-element and I-ordered data, you can enter only the I-index. For IJ-ordered data, you can enter both I- and J-indices. For IJK-ordered data, you can enter I-, J-, and K-indices.)
5. Click Do Probe to perform the probe. The Probe dialog appears.

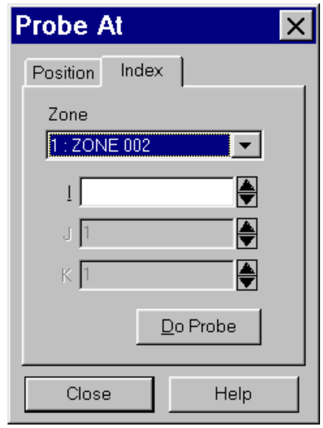



**Figure 26-3.** Index page of the Probe At dialog.

If you have already probed one point, you can specify new indices by increasing or decreasing the displayed values using the up and down arrows at the right of each index field. Doing this automatically performs the probe; you need not click Do Probe again.

## 26.4. Viewing Probed Data from Field Plots

You view probed data in the Probe dialog. The Probe dialog has two pages:

- **Var Values:** Examine values of all variables at any selected location.
- **Zone/Cell Info:** Report characteristics of any location in a data field. The characteristics reported include the indices of the selected cell or point, the zone number, the dimensions of the zone, and the type of zone (ordered or finite-element).

### 26.4.1. Viewing Variable Values

The Var Values page of the Probe dialog, shown in Figure 26-1, lists every variable in the current data set, together with its value at the specified probe point. By default, each variable is shown on a single line, which allows display of about the first ten characters of the variable name and seven significant digits of the variable value.

To display longer variable names or see more digits of the value, deselect the check box labeled One Line per Variable. If there are more variables than will fit in one window, use the Scroll Up and Scroll Down buttons. The Var Values page also displays the zone name and number.

### 26.4.2. Viewing Zone and Cell Info

The Zone/Cell Info page of the Probe dialog, shown in Figure 26-1, lists the following information about any probed data point, regardless of the format of the data:

- The number and name of the probed zone.
- The format of the zone, either ordered or one of the finite-element formats:
  - **FE-Triangle.**
  - **FE-Quad.**
  - **FE-Tetra.**
  - **FE-Brick.**

For ordered zones, the following additional information is displayed:

- The maximum I-, J-, and K-indices of the zone. JMax is 1 for I-ordered data, and KMax is one for I-ordered and IJ-ordered data.
- The type of plane (I, J, or K) which was probed. (For I-ordered and IJ-ordered data, this is always K.)
- In Interpolate mode, the I-, J-, and K-indices of the principal data point of the cell containing the probed point.
- In Nearest Point mode, the I-, J-, and K-indices of the nearest point to the probed point.

For finite-element zones, the following additional information is displayed:

- The total number of points in the zone.
- The total number of elements (cells) in the zone.
- The number of the probed node. This field is filled in only if the point is probed in Nearest Point mode.
- The number of the probed element.
- The number of each node of the probed cell. There are three nodes for FE-surface triangle zones, four nodes for FE-surface quadrilateral and FE-volume tetrahedral zones, and eight nodes for FE-volume brick zones.

## 26.5. Probing XY-Plots

You may probe XY-plots in much the same way you probe field plots. You can use the probe mouse mode to obtain interpolated variable values at any given location, or obtain exact values from a specified (X,Y) data point. There is a significant difference, however. When you probe in the standard mode, Tecplot displays a vertical or horizontal line, depending on whether you are probing along the X- or Y-axis. The probe is performed along the displayed line. All dependent variable values of the active XY-mappings that lie along the probe line are interpolated

and displayed. When you hold down the Ctrl key to enter the Nearest Point mode, the displayed line disappears. Now the exact location of the pointer is important. When you Ctrl-click, Tecplot displays the exact X- and Y-values of the data point closest to the pointer.

**Note:** When probing in XY frame mode, keep in mind whether you are in nearest point or interpolate mode. The presence or absence of the XORed line should indicate the current mode.The exact position of the mouse pointer, while relatively unimportant in interpolate mode, is significant in determining the nearest point.

## 26.5.1. Probing XY-Plots with a Mouse

Just as for field plots, there are two distinct methods of probing an XY-plot with the mouse: probing in Interpolate mode and probing in Nearest Point mode.

The Interpolate mode enables you to click along an X- or Y-axis with the mouse to specify the value of the independent variable, and then view the corresponding dependent values for all active XY-mappings, interpolated to the probed location. Thus, if you are probing along the X-axis, active mappings having Y as the dependent variable are probed. If you are probing along the Y-axis, active mappings having X as the dependent variable are probed. By default, Y is the dependent variable. You can change the dependency for any XY-mapping by modifying the Depend Variable attribute in the Curve-Fit Attributes dialog.

The Nearest Point mode enables you to click at a specific XY-location and view the exact X- and Y-values, along with a variety of information about the XY-mapping, for the data point closest on the screen to the probed location. In Nearest Point mode, probing is independent of whether you are probing along the X- or Y-axis.

**26.5.1.1. Probing XY-Plots in Interpolate Mode.** Interpolate mode is the standard probe mouse mode in XY-plots just as for field plots. You can probe along any of Tecplot's five X-axes, or along any of Tecplot's five Y-axes. By default, probing is performed along the X1 axis.

To enter the Probe Interpolate mode:

1. Choose the Probe tool, indicated by , from the sidebar.

2. Move the pointer into the workspace, where it becomes a cross-hair. When you move into the axis grid area, the cross-hair is augmented by a vertical or horizontal line, depending on whether you are probing along the X-axis or the Y-axis. Click at the desired X- or Y-location. A cross appears on each probed XY-mapping at the probed location, and the Probe dialog appears with a title of Interpolated Values, as in Figure 26-4.
3. Read the desired information from the Probe dialog.

Repeat steps 2 and 3 as desired.
Figure 26-5 shows a workspace with an XY-plot being probed along the X-axis.

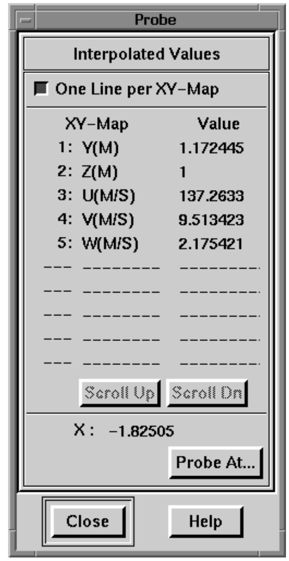



**Figure 26-4.** The Probe dialog for XY-mappings.

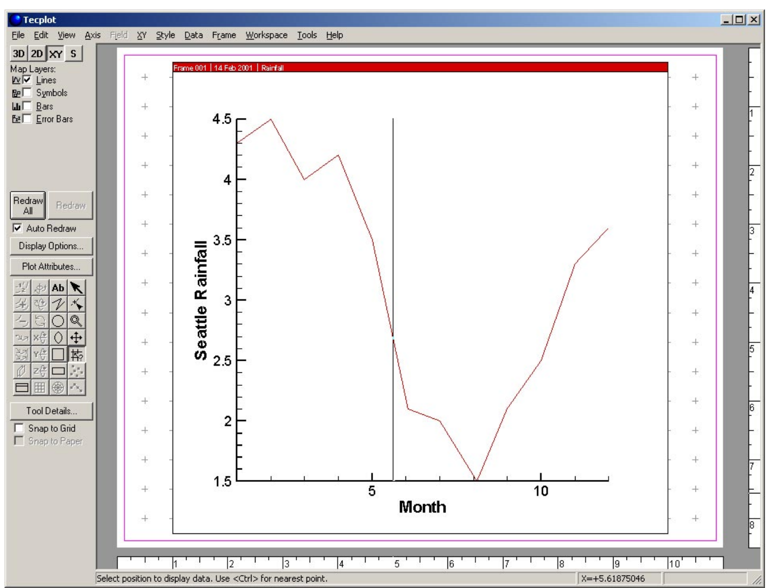



**Figure 26-5.** Probing an XY-plot along the X-axis.

To specify which axis to probe along:

1. Choose the Probe tool from the sidebar.
2. Click Tool Details on the sidebar. The Probe At dialog appears as shown in Figure 26-6.


**Figure 26-6.** The Probe At dialog for XY-plots.

3. Click the button labeled with the name of the axis you want to probe along. If none of the XY-mappings use that axis for the independent variable, the Probe will not return interpolated values.

In the Probe dialog, the probe value is dashed (---) if the probe was out of range for the XY-map.

The probe value is blank if you are probing the XY-map's dependent-variable axis. For example, you are probing an X-axis and the XY-map uses the Y-axis for its independent variable, that is, its function dependency is $x = f(y)$.

The probe value is inactive if the XY-map is not assigned to the specific axis which you are probing. For example you are probing the X1 axis and the XY-map is assigned to the X2 axis. To change the axis you are probing, use the Probe At dialog as specified above.

**26.5.1.2. Probing XY-Plots in Nearest Point Mode.** The Nearest Point probe mode provides the exact X- and Y-values of the data point closest on the screen to the probed location, together with information on the XY-mapping and the zone to which the probed point belongs. If a data point is common to multiple mappings, the probe returns information on the highest numbered mapping. For example, if a data point is plotted as part of two mappings, numbered 1 and 2, the probe results are displayed for mapping 2.

To enter the Probe Nearest Point mode:

1. Choose the Probe tool, indicated , from the sidebar.
2. Move the pointer into the workspace, where it becomes a cross-hair. When you move into the axis grid area, the cross-hair is augmented by a vertical or horizontal line, depending on whether you are probing along the X- or Y-axis. Press and hold down the Ctrl key to see only the cross-hair as you move the mouse. Nearest Point probing is independent of the axis you were probing along.
3. Ctrl-click at the desired probe location. Remember, the nearest point is calculated from the actual location of the cross-hair. (The vertical or horizontal line actually disappears when you press Ctrl. If you intend to probe for several nearest-points, hold down the Ctrl key as you move the cross-hair. This helps you remember that you are probing the point nearest the cursor.)

   A cross appears at the probed location, and the Probe dialog appears with the title Specific Values, as shown in Figure 26-7.

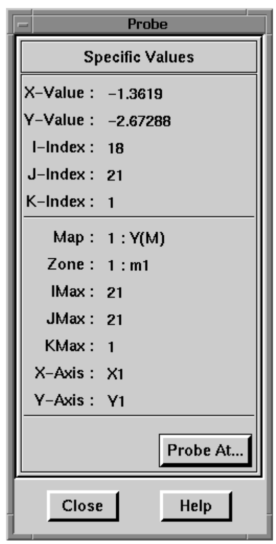



**Figure 26-7.** The Probe dialog for XY-plots (Nearest Point mode).

4. Read the desired data from the Probe dialog.
5. Repeat steps 3 and 4 as often as desired.

## 26.6. Probing XY-Data by Specifying Coordinates and Indices

If you want precise control over your probe location, or if you want to probe using I-, J-, and K-indices, you need to use the Probe At dialog rather than the Probe mouse mode to specify the probe location. You can launch the Probe At dialog from the Data menu (by choosing Probe At), from the Probe dialog (using the Probe At button), or by clicking the Details button while in Probe mouse mode.

To probe at a specified location using position (in Interpolate mode):

1. Launch the Probe At dialog. The Probe At dialog appears, ready for you to enter an X-coordinate. By default the probe is done along the X1 axis.
2. To probe along a different axis, click the button labeled with the name of the axis you want.
3. Enter the exact X- or Y-coordinate of the desired probe location.
4. Click Do Probe to perform the probe. The Probe dialog appears.

To probe at a specified location using data set indices (in Nearest Point mode):

1. Launch the Probe At dialog. The Probe At dialog appears.
2. Click Index to bring up the Index page of the Probe At dialog, shown in Figure 26-8

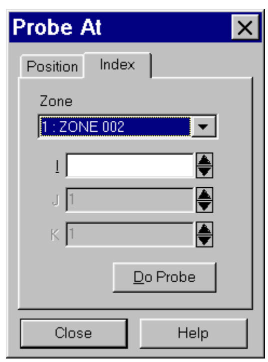



**Figure 26-8.** The Index page of the Probe At dialog for XY-mappings.

3. Select the desired XY-mapping from the XY-Map drop-down.
4. Enter the I-, J-, and K-indices of the desired probe location. (For finite-element and I-ordered data, you can enter only the I-index. For IJ-ordered data, you can enter both I- and J-indices. For IJK-ordered data, you can enter I-, J-, and K-indices.)

5. Click Do Probe to perform the probe. The Probe dialog appears.

If you have already probed one point, you can specify new indices by increasing or decreasing the displayed values using the up and down arrows at the right of each index field. Doing this automatically performs the probe; you do not need to click Do Probe again. If you choose a combination of I-, J-, and K-indices that is valid for the data set but not included in the current mapping, the correct values are returned. No cross appears on the screen because that portion of the data is not being plotted.

# 26.7. Viewing XY Probe Data

The Probe dialog for XY-plots has two different forms, depending on whether you are probing for the interpolated value of the dependent variable or for information on the nearest data point.

## 26.7.1. Viewing Interpolated XY Probe Data

For interpolated plot values, the Probe dialog appears with the heading Interpolated Values and lists every XY-map currently active for the current frame, together with the value of the dependent variable at the specified probe point (see Figure 26-4).

In the Probe dialog, the probe value is dashed (---) if the probe was out of range for the XY-map.

The probe value is blank if you are probing the XY-map's dependent-variable axis. For example, you are probing an X-axis and the XY-map uses the Y-axis for its independent variable, that is, its function dependency is $x = f(y)$.

The probe value is gray (insensitive) if the XY-map is not assigned to the specific axis which you are probing. For example you are probing the X1 axis and the XY-map is assigned to the X2 axis. To change the axis you are probing, use the Probe At dialog as specified above.

By default, each map is shown on a single line, which allows display of about the first ten characters of the map name and seven significant digits of the variable value.

To display longer map names or see more digits of the value, deselect the check box labeled One Line per XY-Map. If there are more XY-mappings than will fit in one window, use the Scroll Up and Scroll Dn buttons. Below the list of XY-maps, the X- or Y-position of the probe is listed.

## 26.7.2. Viewing Nearest Point XY Probe Data

In Nearest Point mode, the Probe dialog appears with the heading Specific Values (see Figure 26-7). You obtain this version of the Probe dialog if you use Probe At Index to specify a

probe position, or if you use the Probe tool and Control-click to specify the probe position. This form displays the following information about the nearest data point to the probed position:

- X-value.
- Y-value.
- I-index.
- J-index.
- K-index.
- The number and name of the XY-mapping associated with the data point.
- The number and name of the zone referenced in the XY-mapping.
- The maximum I-index of the zone.
- The maximum J-index of the zone.
- The maximum K-index of the zone.
- The X-axis associated with the XY-mapping.
- The Y-axis associated with the XY-mapping.

## 26.8. Probing to Edit

Using the Adjustor tool, you can probe and edit specific data points. In Adjuster mode, you can actually modify the X- and Y-coordinates of your data with the mouse. To avoid inadvertent changes, data point editing must be specifically enabled before you can actually change points with the Adjustor.

You can edit data points either by moving them with the mouse (in XY- and 2-D plots only), or by using the Probe/Edit Data dialog to enter new values for any variable in the probed data point.

- **To enable data point editing:** In the Edit menu, select Allow Data Point Adjustment. On Motif systems, a small box is displayed next to the menu item to indicate the option is active. On Windows systems, a check mark appears next to the menu item to indicate the option is active.
- **To disable data point editing:** In the Edit menu, select Allow Data Point Adjustment again. The mark indicating the option is active disappears.

### 26.8.1. Editing Data with the Mouse

In XY- and 2-D plots, you can select and move data points with the Adjustor mouse mode. You can select multiple data points and move them as a group. When you move data points with the mouse, you will not actually see the changes until you redraw the screen.

To edit your data with the mouse:

1. On the sidebar, choose the Adjustor tool, indicated by .
2. Move the pointer into the workspace, where it becomes the Adjustor.
3. Enable data point editing, if you have not already done so.
4. Click on a single point to select it. In the Adjustor mode, you must be within one-half of the selection handle's width to select the data point. To select multiple points, you can either Shift-click after selecting your initial point to select additional points, or you can draw a group select band to select the points within the band. (In XY-plots, you can select points from only one mapping at a time.)
5. Once you have selected all desired points, move the Adjustor over the selection handles of one of the points, then click-and-drag to the desired location of the chosen data point. Other selected points will move as a unit with the chosen data point, maintaining their relative positions.

   For XY-plots, if several mappings are using the same data for one of the variables, adjusting one of the mappings will result in simultaneous adjustments to the others. You can avoid this by pressing the H or V keys on your keyboard while adjusting the selected point. The H and V keys restrict the adjustment to the horizontal and vertical directions, respectively.

### 26.8.2. Editing Data with the Probe/Edit Data Dialog

To probe to edit using the Probe/Edit Data dialog:

1. On the sidebar, choose the Adjustor tool, indicated by the  button.
2. Move the pointer into the workspace, where it becomes the Adjustor.
3. Enable data point editing, if you have not already done so.
4. Double-click on the point you want to edit, or click on the point and then click Object Details on the sidebar. The Probe/Edit Data dialog appears, as shown in Figure 26-9. All variables in the zone or mapping are listed, along with their values at the probed point. In 2D and 3D frame mode, the Probe/Edit Data dialog has Scroll Up and Scroll Dn buttons which are active if the data set has more variables than can be displayed on one page of the dialog.

   **Note:** If you attempt to double-click, but move the mouse between clicks, you may find that you have accidentally moved your data point.

**5.** Enter new values as desired.

The lower half of the Probe/Edit Data dialog is a copy of the Probe At dialog's Index page. You can use this area to specify a new zone or mapping to probe, along with the specific points to probe and edit. Thus, for example, you can edit one point, then increase or decrease the displayed indices to edit the next point along a mapping.

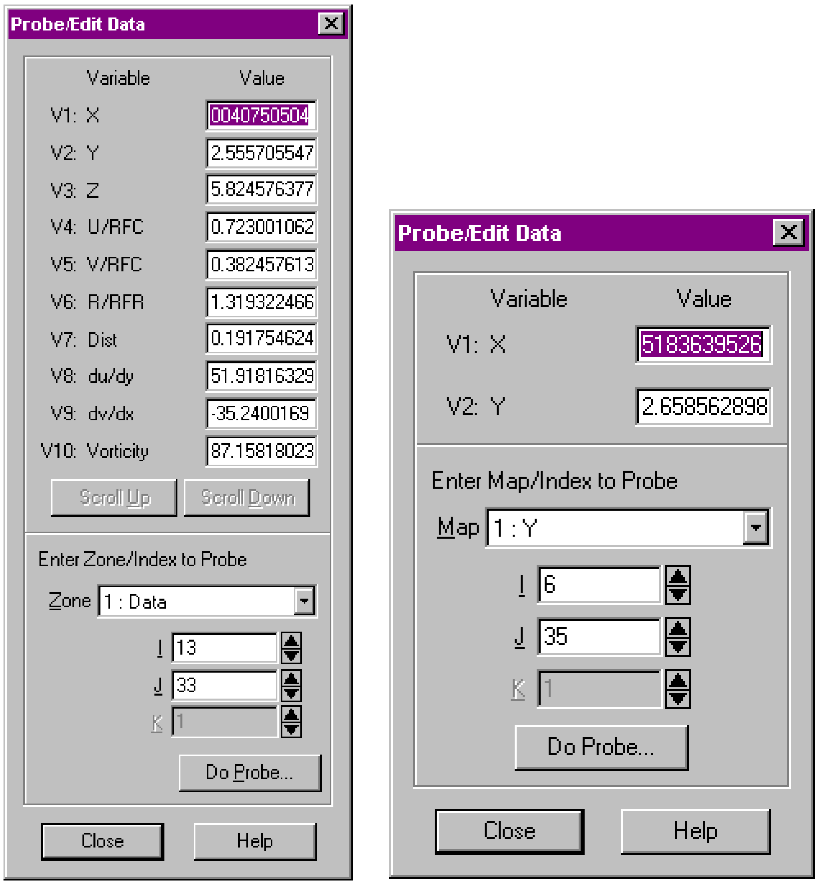



**Figure 26-9.** Probe/Edit Data dialog for field plots (left) and XY-plots (right).

# CHAPTER 27 Blanking

Blanking is the capability of Tecplot to exclude certain portions of zones from being plotted (in other words, selectively display certain cells or data points). In 3-D, the result is analogous to a cutaway view.

## 27.1. Blanking 2- and 3-D Plots

In the following discussions, the term cell is used. In I-ordered data sets, a cell is the connection between two adjacent points. In IJ-ordered data sets, a cell is the quadrilateral area bounded by four neighboring data points. In IJK-ordered data sets, a cell is the six-faced (hexahedral) volume bounded by eight neighboring data points. For finite-element data sets, a cell is equivalent to an element.

There are three forms of blanking, as follows:

- **Value-Blanking:** Cells of all zones or XY-mappings are excluded based on the value of a variable (the value-blanking variable) at the data point of each cell or at the point where each cell intersects with a constraint boundary depending on the type of value blanking applied.
- **IJK-Blanking:** Cells of one IJK-ordered zone are included or excluded based on the index values.
- **Depth-Blanking:** Cells in a 3-D plot are visually excluded based on their distance from the viewer plane.

All types of blanking affect all field layers, except the Boundary zone layer. Value-blanking and IJK-blanking affect data operations such as streamtrace extraction, iso-surface extraction, slicing, and so on. Blanking is not performed in 3D frame mode when wire-frame sketches are drawn while rotating, translating, slicing, and so on.

Blanking on volume zones may produce different results, depending upon the Surfaces to plot setting on the Volume Attributes dialog. See Section 20.1, “Choosing Which Surfaces to Plot,”

for more details.

All types of blanking may be used in a single plot. They are cumulative: cells blanked from any of the options do not appear.

Value-blanking and depth-blanking affect all zones of all types of data, while IJK-blanking affects only one IJK-ordered zone. IJK-blanking is available in 2D or 3D frame mode; value-blanking is available in XY, 2D, or 3D frame mode; depth-blanking is only available in 3D frame mode.

### 27.1.1. Value-Blanking 2- and 3-D Plots

Value-blanking allows you to selectively eliminate or trim cells and elements from XY-, 2-D, and 3-D field plots. The two forms of value-blanking are referred to as whole cell and precise blanking. The whole cell or precise blanking of cells is based on one or more user-defined constraints. For each active constraint you specify a value-blanking variable, a constant value or another variable, and a conditional statement telling Tecplot that region to blank in relation to the specified variable or constant. Whole cell blanking eliminates entire cells and therefore can result in a jagged blanking boundary, while precise blanking trims the display of data along a constraint boundary that you specify (precise blanking is only available in 2D frame mode).

**27.1.1.1. Whole Cell Blanking.** To use whole cell blanking:

1. From the Style menu, choose Value-Blanking. The Value-Blanking dialog appears as in Figure 27-1.
2. Select the Include Value-Blanking check box. Value-blanking has no effect until this check box is turned on and at least one constraint is activated (see step 4), regardless of the settings of the other value-blanking parameters.

   Figure 27-2 demonstrates the various effects of whole cell and precise value-blanking modes.
3. Choose the Blank entire cells option. Specify how the cell is blanked by selecting one of three selections from the option menu:
   - **All corners are blanked:** Cells are removed from the plot if all of their data points satisfy one or more of the active blanking constraints.
   - **Any corner is blanked:** Cells are removed from the plot if any of their data points satisfy one or more of the active blanking constraints.
   - **Primary corner is blanked:** Cells are removed from the plot based on the value at the primary corner of each cell. For ordered zones, the primary corner of a cell is the data point with the smallest indices of that cell. For finite-element zones, the primary corner is the first node listed in the data file’s connectivity list for that element.

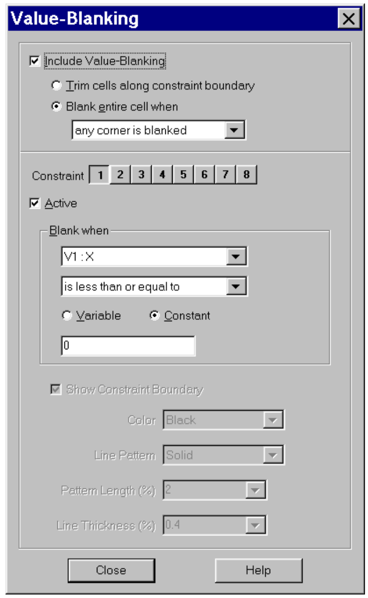



**Figure 27-1.** The Value-Blanking dialog when frame mode is 2D or 3D.

4. Activate a constraint by selecting the Active check box associated with the desired constraint number.
5. For any active constraint select the variable to use for value-blanking. This man be any variable, even one that is being used elsewhere in the plot. It is often convenient to create a new variable for use as the value-blanking variable. In this way you can manipulate its values without changing any other part of the plot. If no value-blanking variable is available, you can create one using the Specify Equations dialog, accessed from the Alter option of the Data drop-down menu (for instance, "{VBlank}=1"). See Section 25.1.1, “Equation Syntax.”
6. Specify one of the following operations to describe how the blanking variable will be compared to the constant or variable following it:
   - **Is less than or equal to**: Cells for which the value-blanking variable has a value less than or equal to the specified constant or variable are removed from the plot.
   - **Is greater than or equal to**: Cells for which the value-blanking variable has a value greater than or equal to the specified constant or variable are removed from the plot.
7. Specify the value-blanking constant or variable used for comparison with the value-blanking variable:
   - **Variable**: This is the variable that is compared against the value-blanking variable to determine which cells are removed from the plot.

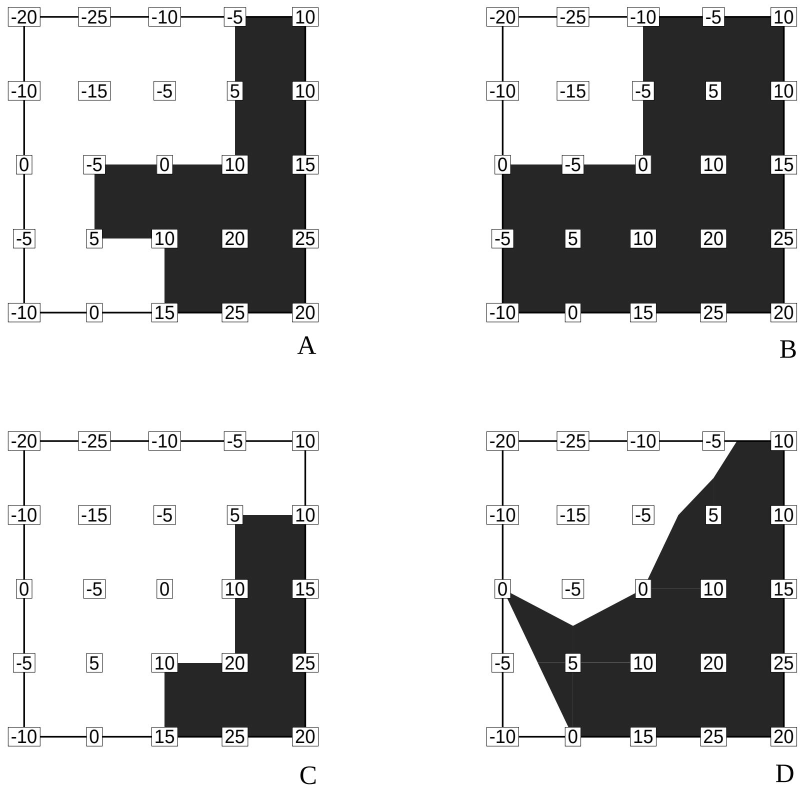



**Figure 27-2.** The effects of the different value-blanking options for a constraint where a variable is less than or equal to zero. The dark shading indicates the areas which are not blanked. A) Blank cell when primary corner is blanked. B) Blank cell when all corners are blanked. C) Blank cell when any corner is blanked. D) Trim cells along mathematical constraint boundary.

- **Constant:** This is the number that is compared against the value-blanking variable to determine which cells are removed from the plot.

The Show Constraint Boundary check box will show you the line which separates the region of your data which is blanked from the region which is not blanked. Value-blanking has no effect on boundaries of an ordered zone. If the boundary is turned on, the boundary of the entire zone (without value-blanking) is plotted.

For finite-element data, value-blanking can affect the view of previously extracted boundaries, because each extracted boundary is a zone (see Section 20.4., "Extracting Boundaries of Finite-Element Zones" ).

Figure 27-3 shows an IJ-ordered mesh with whole cell and precise value-blanking in use.

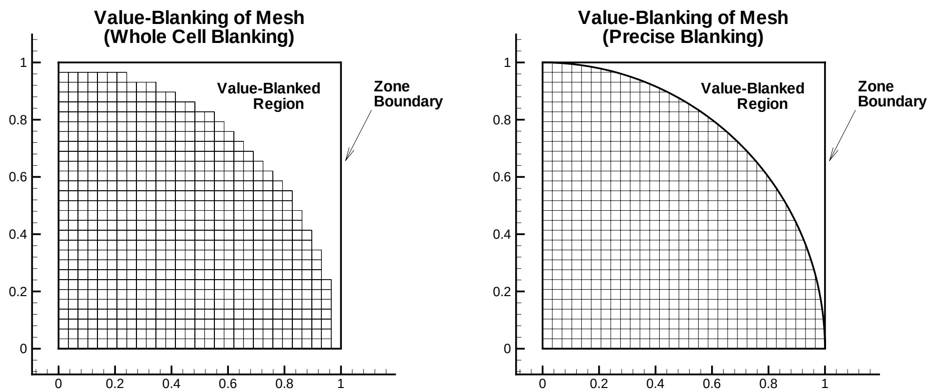



**Figure 27-3.** An IJ-ordered mesh with whole cell and precise value-blanking.

**27.1.1.2. Precise Blanking.** Precise blanking is only available in 2D frame mode. To use precise blanking:

1. From the Style menu, choose Value-Blanking. The Value-Blanking dialog appears as in Figure 27-4.
2. Select the Include Value-Blanking check box. Value-blanking has no effect until it is turned on and at least one constraint is activated (see step 4), regardless of the settings of the other value-blanking parameters.

   Figure 27-2 demonstrates the various effects of whole cell and precise value blanking modes.
3. Choose the Trim Cells along Constraint Boundary option.
4. Activate a constraint by selecting the Active check box associated with the desired constraint number.
5. For any active constraint select the variable to use for value-blanking. This variable can be any variable, even one that is being used elsewhere in the plot. It is often convenient to create a new variable for use as the value-blanking variable. In this way you can manipulate its values without changing any other part of the plot. If no value-blanking variable is available, you can create one using the Specify Equations dialog (for example, "{VBlank}=1"). See Section 25.1.1, "Equation Syntax."

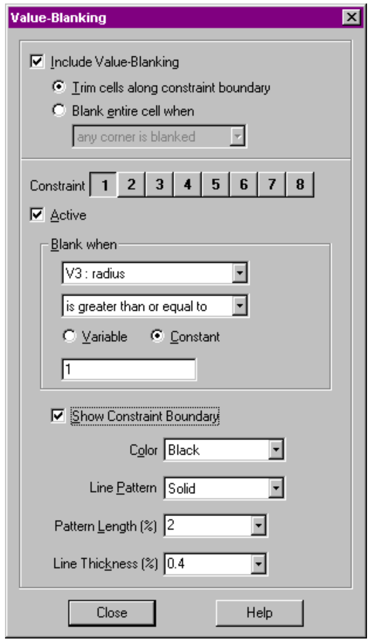



**Figure 27-4.** The Value-Blanking dialog in 2D and 3D frame mode, set for precise blanking.

6. Specify one of the following operations to describe how the blanking variable will be compared to the constant or variable following it:
   - **Is less than or equal to:** Trim away all regions where the value-blanking variable is less than or equal to the specified constant.
   - **Is greater than or equal to:** Trim away all regions where the value-blanking variable has a value greater than or equal to the specified constant or variable.
7. Specify the value-blanking constant or variable used for comparison with the value-blanking variable:
   - **Variable:** This is the variable that is compared against the value-blanking variable.
   - **Constant:** This is the number that is compared against the value-blanking variable.
8. Select the Show Constraint Boundary check box to control the visibility of the defined constraint boundary line. The various attributes of the constraint line such as color, line pattern, pattern length, and line thickness can also be set to produce the desired effect.

Value-blanking has no effect on boundaries of an ordered zone. If the boundary is turned on, the boundary of the entire zone (without value-blanking) is plotted. For finite-element data,

value-blanking can affect the boundary extraction. For more information see Section 16.4, "Extracting Boundaries of Finite-Element Zones."

Figure 27-3 shows a mesh with whole cell and precise value-blanking in use. Figure 27-5 demonstrates a more complex usage of precise value-blanking by overlaying multiple frames, each using some of the same constraints.

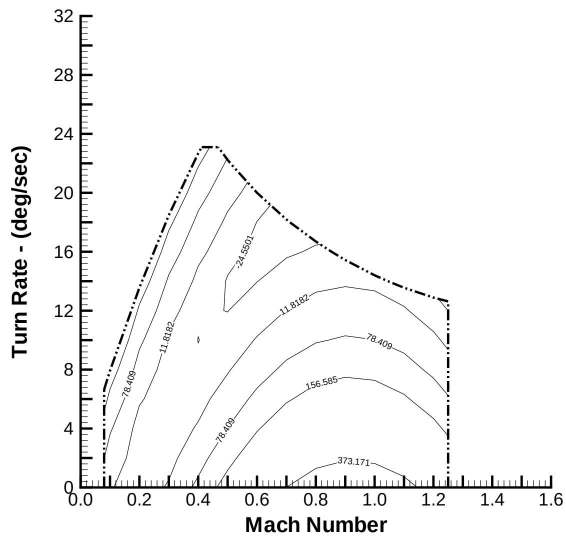



**Figure 27-5.** A single contour variable with precise blanking.

### 27.1.2. IJK-Blanking

IJK-blanking is available only for 3-D volume zones. IJK-blanking removes a selected portion of one IJK-ordered zone from the plot. This allows you to create cutaway plots, plots showing the exterior of some data set with a section "cut away" to show the interior, such as the plot shown in Figure 27-6. You define the blank region by specifying the following:

- The IJK-ordered zone in which the blanking is to be performed.
- I-, J-, and K-index ranges for the blank region, either using specific index values or percentages of the index range.
- Whether Tecplot should blank the interior or exterior of the defined region.

To use IJK-blanking, you must have an IJK-ordered zone, and the current frame must be in 2D or 3D frame mode. Unlike value-blanking, which operates on all zones within a single frame,

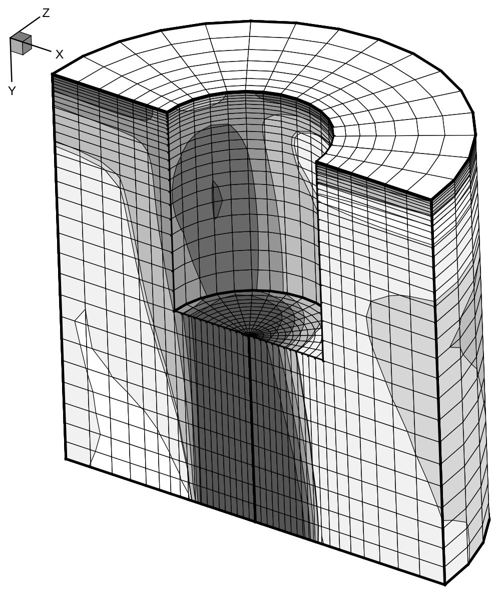



**Figure 27-6.** A cutaway plot created with IJK-blanking.

IJK-blanking can only be used on a single zone within a frame, and the zone must be IJK-ordered.

To use IJK-blanking:

1. From the Style menu, choose IJK-Blanking. The IJK-Blanking dialog appears as in Figure 27-7.
2. Select the Include IJK-Blanking check box to turn on IJK-blanking for the current frame. IJK-blanking does not take effect until this option is turned on, nor are any of the other controls sensitive.
3. Specify the domain of the IJK-blanking by choosing one of the following options:
   - **Interior:** Cells within the specified index ranges are blanked. Those outside are plotted. This creates a ‘‘hole’’ in the zone. The left side of Figure 27-8 shows an ordered zone with IJK-blanking with Interior domain.
   - **Exterior:** Cells outside the specified index ranges are blanked. Those inside are plotted. This plots a sub-zone of the zone. The right side of Figure 27-8 shows an ordered zone with IJK-blanking with Exterior domain.

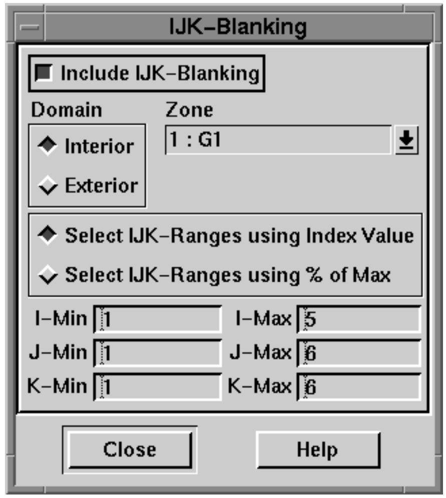



**Figure 27-7.** The IJK-Blanking dialog.

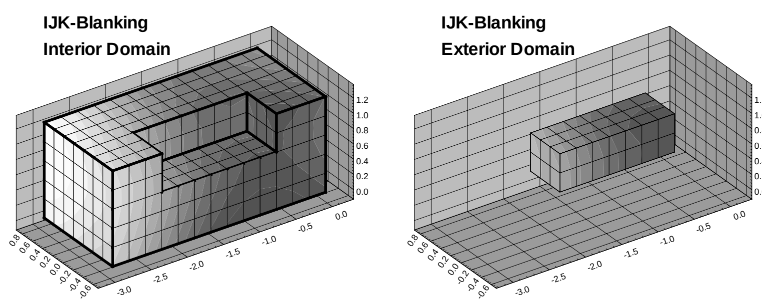



**Figure 27-8.** IJK-blanking with Interior domain (left) and Exterior domain (right).

4. Select the zone to which IJK-blanking is applied from the Zone drop-down. The zone must be IJK-ordered. You may select only one zone at a time.
5. Specify the format in which you will specify the index ranges by selecting one of the following option buttons:
   - **Select IJK-Ranges Using Index Values:** If you select this option, you specify the I-, J-, and K-index ranges using actual minimum and maximum indices.

- **Select IJK-Ranges Using % of Max:** If you select this option, you specify the I-, J-, and K-index ranges as start and end percentages of the maximum index. For example, you could blank the middle third of a data set by setting the start percentage to 33.3 and the end percentage to 66.6.

6. Enter the I-, J-, and K-index ranges in the fields provided.

When you save a layout, macro, or stylesheet, the IJK-blanking index ranges are stored as the percentage of the maximum index regardless of how you chose to enter them. This way, one file can be used for different zone sizes.

### 27.1.3. Cutaway Plots

Cutaway plots are plots of a 3-D volume zone in which a portion of the zone is blanked using IJK-blanking so that the interior of the zone can be seen. Create Figure 27-6 as follows:

1. Create an IJK-ordered zone, and create a 3-D contour plot of the zone. (Leave the Mesh zone layer turned on.)
2. Set the contour plot type to either Flood or Both Lines and Flood.
3. Use the IJK-Blanking dialog to blank out the appropriate region. Use the Interior blanking domain.

A more complex cutaway plot is shown in Figure 27-9. This plot contains iso-surfaces inside the cutout region.

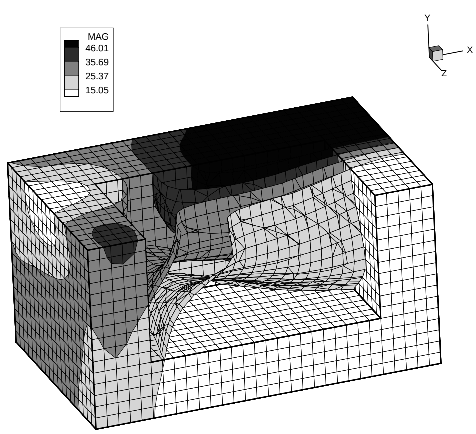



**Figure 27-9.** Cutaway plot with iso-surfaces inside cutout region.

To create the plot in Figure 27-9, continue with the following steps:

1. Set the contour plot type of the IJK-ordered zone to Both Lines and Flood.
2. Set the IJK-mode of the IJK-ordered zone to Volume.
3. Use the IJK-Blanking dialog to blank out the appropriate region, and choose the Exterior blanking domain. You should now have a plot of the region you want to blank out.
4. Call up the 3D Iso-Surface Details dialog from the Field menu. Select Show Iso-Surfaces and set the Draw Iso-Surfaces At drop-down menu to Each Contour Level.
5. From the Data menu, choose Extract, then choose Iso-Surfaces to extract the contour iso-surfaces of the region you want to blank out.
6. Change the contour plot type of the new iso-surface zones to Flood.
7. Change the IJK-blanking domain to Interior. This changes the IJK-blanking to plot the entire zone minus the blanked region.
8. Set the IJK mode of the IJK-ordered zone to Face.

The iso-surfaces appear in the plot because they were extracted as separate zones.

### 27.1.4. Depth-Blanking

Depth-blanking removes cells in a 3-D plot based upon how close or far they appear from the screen. The options below are available on the Depth Blanking dialog, shown in Figure 27-10.

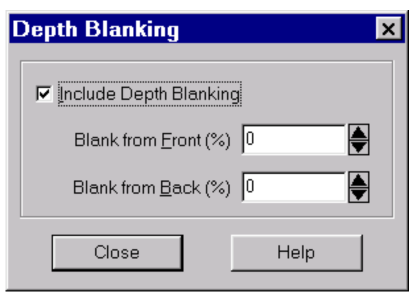



**Figure 27-10.** The Depth Blanking dialog.

- **Include Depth Blanking:** Select this check box to toggle depth-blanking on and off.
- **Blank from Front (%):** Blank cells appearing closer to the viewer than this plane. The value entered is the plane position in percentage of depth from the closest corner of the bounding box of the data to the furthest corner of the bounding box.

At the default of zero, the plane is at the depth of the closest corner of the bounding box. No cells on the front of the plot are blanked. At 50, the front half of the plot will be blanked. In particular, cells closer to the viewer than the front of the blanking plan, and cells further from the viewer than the blanking plane, may be blanked.

- **Blank from Back (%):** Blank cells appearing farther from the viewer than this plane. The value entered is the plane position in percentage of depth from the furthest corner of the bounding box of the data to the closest corner of the bounding box.

  At the default of zero, this plane is at the depth of the furthest corner of the bounding box. No cells on the back of the plot will be blanked. At 50, the back half of the plot will be blanked.

## 27.2. Blanking XY-Plots

For XY-plots, blanking is the capability of Tecplot to exclude data points from consideration in the resulting plot. On a global scale, only value-blanking is available. To plot specific index ranges you can use the Index Attributes option from the XY menu to limit index ranges per XY-mapping. The Curves option from the XY menu can provide another form of blanking, by allowing you to limit the range for the independent variable for individual XY-mappings.

Figure 27-11 shows two plots. The original data for the plots contain some “bad” data points. The bad data points were identified as those with a Y-value greater than 0.6. The plot on the left uses all data points, including the bad data points, to draw a curve. The plot on the right has filtered out the bad data points by using value-blanking where all points are removed if Y > 0.6. Blanking does not necessarily have to be on the independent or dependent variable.

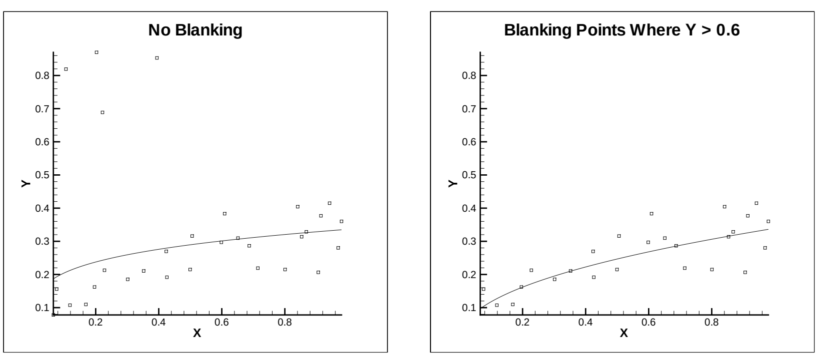



**Figure 27-11.** XY-plots showing the effect of value-blanking.

# CHAPTER 28 Using Macros

Macro files allow you to automate Tecplot. Macro files contain a sequence of macro commands, and may contain macro function definitions. Macro functions act like macros-within-macros: they allow you to combine macro commands that you frequently use into a single unit callable from within another macro. This chapter focuses on the Tecplot menu options for recording and playing back macros. The *Tecplot Reference Manual* describes the Tecplot macro language in detail.

Macros are very useful for performing repetitive operations such as setting up frames, reading in data files and layout files, manipulating data, and creating plots. They are also necessary for running Tecplot in batch mode. See Chapter 29, “Batch Processing.”

The Macro sub-menu, found under the File menu, provides the following control options:

- **Play:** Calls up the Load/Play Macro File dialog, to select a macro file to load and play.
- **View:** Select this option to call up the Macro Viewer dialog, which provides several command buttons for stepping through and debugging a macro file.
- **Record:** Calls up the Macro Recorder dialog, which provides several command buttons for recording a series of actions to a macro file for playing back at a later time.

## 28.1. Creating Macros

The simplest way to create a macro is to have Tecplot record it for you. You can then use any ASCII text editor to edit the macro file. While editing your file, you can, for example, add macro function definitions, or add loops and other control commands.

Tecplot’s Macro Recorder lets you record a macro as you perform a sequence of actions interactively to obtain precisely the results you want. After recording your macro, you can edit it with an ASCII text editor to remove redundant operations, compress repetitive actions into loops, and otherwise modify the macro. Using the Macro Recorder is the quickest and surest way to become familiar with the Tecplot macro language.

To record a macro with the Macro Recorder:

1. From the File menu, choose Macro, then choose Record. The Write Macro File dialog is displayed.
2. Specify a macro file name.
3. Click OK to initiate the recording of the macro file. The Macro Recorder dialog now appears, as shown in Figure 28-1. This dialog must remain up during the recording session.

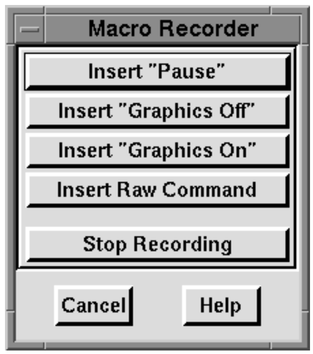



**Figure 28-1.** The Macro Recorder dialog.

4. Perform the actions you want recorded using the Tecplot interface.
5. Click Stop Recording on the Macro Recorder dialog when you are finished with the sequence of actions you want recorded.

While recording macros, you can use any of the following four buttons on the Macro Recorder dialog to add specific macro commands to your macro:

- **Insert "Pause":** Adds a "pause" command to the macro. When you play a macro including a pause command, Tecplot displays a message box when it reaches the pause command, and waits for you to click OK before continuing to process the macro.
- **Insert "Graphics Off":** Adds a "graphics off" command to the macro. When you play a macro containing a "graphics off" command, Tecplot stops displaying graphics in the workspace from the "graphics off" command until a "graphics on" command is encountered.
- **Insert "Graphics On":** Adds a "graphics on" command to the macro.
- **Insert Raw Command:** Brings up a dialog in which you can enter any valid Tecplot macro command. For example, you can add "`$!LOOP 10`" at the start of a section you want to repeat 10 times, then "`$!ENDLOOP`" at the end. See the *Tecplot Reference Manual* for information on the Tecplot macro language.

### 28.1.1. Defining Macro Functions

When editing your macros, you can add macro function definitions and macro function calls. Macro functions have the following form:

```
$!MACROFUNCTION
  NAME = functionname
 .
 .
 .
$!ENDMACROFUNCTION
```

Between **$!MACROFUNCTION** and **$!ENDMACROFUNCTION**, you can include any legal macro command except **$!MACROFUNCTION**. These included macro commands are associated with the *functionname* specified as the value of the **NAME** parameter, but are not executed until the macro function is called with the **$!RUNMACROFUNCTION** macro command.

For example, the following macro function turns on the Contour zone layer, turns off the Mesh zone layer, sets the contour plot type to Both Lines and Flood for zones 1, 2 and 3, then chooses gray scale color mapping:

```
$!MACROFUNCTION
  NAME = "graycontour"
  RETAIN = Yes
$!FIELDLAYERS SHOWCONTOUR = YES
$!FIELDLAYERS SHOWMESH = NO
$!FIELD [1-3]  CONTOUR{CONTOURTYPE = BOTHLINESANDFLOOD}
$!COLORMAP CONTOURCOLORMAP = GRAYSCALE
$!REDRAW
$!ENDMACROFUNCTION
```

The **RETAIN** parameter tells Tecplot to retain the macro function definition for use in subsequent macro calls; this allows you to define a macro function once in some macro you load every time you run Tecplot, and continue to use it throughout your Tecplot session. See Section 28.4, “Doing More with Macros,” for a more sophisticated example of a macro function.

Use the **$!RUNMACROFUNCTION** macro command to call your macro function. For example, to call the “graycontour” macro function defined above, use the following macro command:

```
$!RUNMACROFUNCTION "graycontour"
```

If the macro function requires any parameters, you combine them into a parenthesized list which you give as a second argument to **$!RUNMACROFUNCTION**, as in the following example:

```
$!RUNMACROFUNCTION "Process Plane" (K,|LOOP|)
```

You can use the **$!RUNMACROFUNCTION** command within other macro functions; calls may be nested up to ten deep.

## 28.2. Playing Back Macros

Once you have created a macro file, you have four methods in Tecplot for playing it back.

The four different methods are:

- From the command line.
- From the File menu, under the Macro Play option in the interface.
- Stepping through commands using the Macro Viewer.
- From the Quick Macro Panel.

The following sections explain each of these methods.

### 28.2.1. Preparing to Play Back a Macro

Often, the commands in a macro file rely on Tecplot being in a particular state. It is usually a good practice to have commands at the start of a macro that force Tecplot into a known state, **$!NEWLAYOUT** is a good command to do this since it deletes all data sets and frames and creates a single empty frame with a default size and position.

If you will always run your macro from the command line, then you can be sure Tecplot will be in its initial state when the macro begins processing. Including layout files, data files or stylesheet files on the command line along with the macro file is fine as long as the macro expects them.

If your macro performs some intermediate task, it is up to you to make sure Tecplot is in the same (or a similar) state when you run the macro as the state the macro was designed to start in.

### 28.2.2. Running a Macro From the Command Line

You can immediately play a macro when you first start Tecplot by simply including the name of the macro file on the command line. In Windows, this can be accomplished by either dragging and dropping a macro file onto the Tecplot icon, or by using the command line from an MS-DOS dialog.

For example, to run the macro file **mymacro.mcr** from a UNIX or DOS command line prompt, type:

```
tecplot mymacro.mcr
```

If you name your macro file without the **.mcr** extension you can still run Tecplot with the macro file. However, you must include the **-p** flag on the command line. To run the macro file called **mymacro.mmm** you would type:

```
tecplot -p mymacro.mmm
```

In Windows you cannot drag and drop a macro file onto the Tecplot icon if it does not have the **.mcr** extension. Tecplot will think it is an ASCII data file and attempt to read it in as such.

If you want the macro viewer to automatically appear so you can see the macro commands prior to their execution, you can include the **-z** flag on the command line.

Macros can also be played back in batch mode (i.e., no graphics are displayed). See Chapter 29, “Batch Processing,” for details.

### 28.2.3. Running a Macro From the Interface

You can play a macro from within Tecplot by using the Play option under the File main menu. This plays back the macro file without stopping until it reaches the end of the file.

To play back a macro file from the Tecplot interface:

1. From the File menu, choose Macro, then choose Play. The Load/Play Macro File dialog is displayed.
2. Specify a macro file name.
3. Click OK.

Tecplot immediately starts playing the specified macro file.

### 28.2.4. Running Macros from the Quick Macro Panel

The Quick Macro Panel (Figure 28-2) is Tecplot’s quick access mechanism for storing and retrieving your favorite, commonly used macro functions. This panel allows you to quickly play a macro function by clicking on the button in the panel that is linked to that macro function.

The Quick Macro Panel is linked to a special macro file that contains only macro function definitions. When Tecplot first starts up, it looks for this file under one of the following names, in the following order:

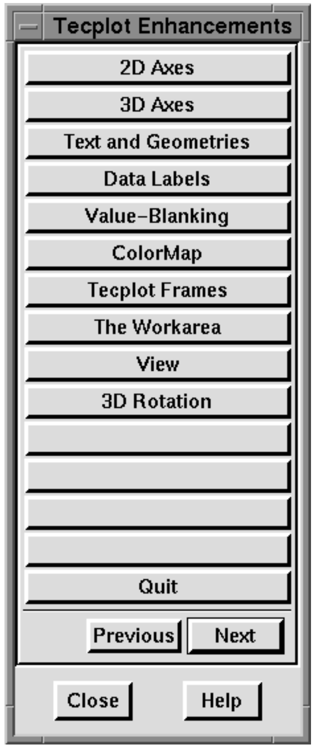



**Figure 28-2.** The Quick Macro Panel.

1. The file **`tecplot.mcr`** in the current directory.
2. The file **`.tecplot.mcr`** in your home directory (UNIX), or **`tecplot.mcr`** in the your home directory (Windows). Under Windows, your home directory is determined by the two environment variables **`HOMEDRIVE`** and **`HOMEPATH`**. If they are not set, Tecplot skips your home directory.
3. The file **`tecplot.mcr`** in the Tecplot home directory.

If Tecplot finds the file, it loads it and associates each button on the Quick Macro Panel with a specific macro function.

You can specify a different Quick Macro file by adding the **`-qm`** option flag in front of the macro file name to the command line.

The following command starts Tecplot and installs the macro functions defined in the file **`myteccmd.mcr`** into the Quick Macro Panel:

```
tecplot -qm myteccmd.mcr
```

If you want Tecplot to call up the Quick Macro Panel immediately after start up, include the **-showpanel** option flag at the end of the command as well.

For example, the following command starts Tecplot and immediately calls up the Quick Macro Panel:

```
tecplot -qm myteccmd.mcr -showpanel
```

To see an example of a macro function file, look at the Quick Macro file **qmp.mcr** located in the examples/mcr sub-directory below the Tecplot home directory.

Once the Quick Macro Panel has been installed you can run a macro by pressing its associated button on the Quick Macro Panel.

### 28.2.5. Linking Macros to Text and Geometries

Each text or geometry you create can be linked to a macro function. This macro function is called whenever the user holds down the control key and clicks the right mouse button on the text or geometry.

For example, if you have pieces of text, each representing a different well, Ctrl-right click on any piece could run a macro that brings up an XY-plot of that well's data.

Macro functions are specified with the "Link to Macro function" field in the Geometry dialog or in the Text Options dialog. If desired, the macro function may be listed with one or more parameters.

## 28.3. Debugging Macros

Use the Macro Viewer to step through and debug your macro file. This dialog allows you to add and delete breakpoints, view and set watch variables, and view state variables local to the macro currently loaded into the Macro Viewer. Selecting View from the Macro sub-menu displays the Macro Viewer dialog, shown in Figure 28-3. Using the Macro Viewer to look at a macro file built with the Macro Recorder is a quick and easy way to explore the Tecplot macro language.

To load a macro file into the Macro Viewer, click Load Macro. This calls up the Load/Play Macro File dialog for you to specify which macro file to load. Macro files typically have the extension **.mcr**.

The specified macro is loaded into the macro viewer for you. If you already had a macro loaded, it is discarded and the new macro is loaded in its place.

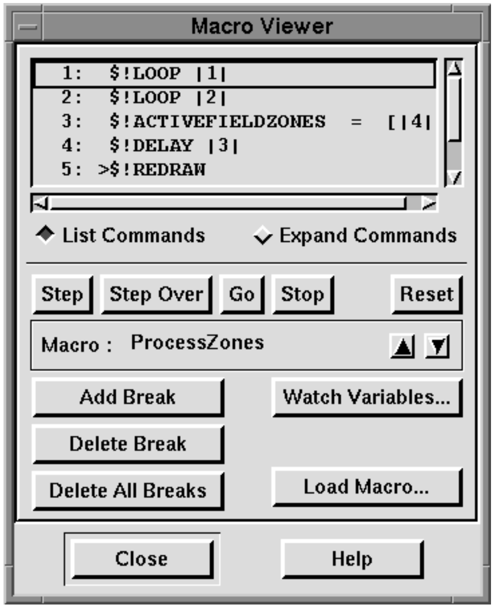



**Figure 28-3.** The Macro Viewer dialog.

The Macro Viewer dialog displays the text of the currently loaded macro file at the top of the dialog. A **>** (greater than) marks the currently active line, that is, the line that Tecplot is about to evaluate. Click Step to evaluate the currently active line. The **>** sign then moves to the next line.

## 28.3.1. Macro Context

In the Macro Viewer dialog, the field labeled Macro displays the name of the macro or macro function you are currently evaluating. In most cases, this field displays the name **`MAIN`**, which means that the macro commands currently shown in the macro text display come from within the main macro body, that is, not from inside a macro function. If the macro you are viewing contains a call to a macro function, then when you evaluate (or step into) that macro function call (when you evaluate a **`$!RUNMACROFUNCTION`** command) the name displayed in the Macro field changes. The new name displayed is the name of the macro function just called. At the same time, the display in the top of the Macro Viewer dialog changes to show the macro function text for the called macro function.

Pressing on the up and down arrows located at the right hand side of the Macro display field shifts the macro context, that is, it lets you move between the text of the called macro function, and the text of the calling macro or macro function. If you switch context to the calling macro

function, the **$!RUNMACROFUNCTION** command that called the macro is displayed with a **^** (caret) in front of it. This helps you quickly determine which command line called the macro function currently under evaluation. The down arrow then moves you back down a level to the called macro you were just viewing.

### 28.3.2. Changing the Macro Command Display Format

Tecplot displays the macro in the viewer in one of two formats: a short format that lists the macro commands, one command per line, and a long format which expands a single, simple macro command to show all of its sub-commands and parameters. The short format is the default for the Macro Viewer.

- To choose the short form, select the List Commands option.
- To choose the long form, select the Expand Commands option.

### 28.3.3. Evaluating a Macro File with the Macro Viewer

The Macro Viewer dialog's main purpose is to allow you to step through a macro's commands in a variety of ways so you can view and debug a macro file. With this dialog you can evaluate each line, including the commands within nested macro function calls, or just have Tecplot run the macro automatically while you watch.

**28.3.3.1. Stepping through a Macro Line by Line.** The main activity you do in the Macro Viewer dialog is evaluate macro commands line by line. The **>** (greater than) marks the currently active command, that is, the command that Tecplot is about to evaluate. It moves to the next command after the currently active command is evaluated.

To evaluate a macro command, click Step. When a **$!RUNMACROFUNCTION** command is encountered, the macro viewer steps into the called function.

Step Over also processes each macro command, line by line however, when a **$!RUNMACRO-FUNCTION** command is encountered the entire function is processed.

You can also view or play a macro all the way through without stopping. To play the macro without stopping after each step, click Go. Tecplot continues until it either receives a stop signal from the Stop button, or it finishes playing the macro, or it encounters a breakpoint. See Section 28.3.4, "Adding and Deleting Breakpoints," on using breakpoints.

To stop the playing of a macro that the Go control started, click Stop.

You can restart a macro from the beginning, so you can evaluate it again from within the Macro Viewer. To restart the evaluation of a macro, click Reset.

**Note:** If your macro assumes Tecplot is in a particular state when it starts processing then you must make sure Tecplot is in this state before you click Reset and start the macro processing.

### 28.3.4. Adding and Deleting Breakpoints

An important debugging feature that the Macro Viewer provides is the ability to add breakpoints within a macro's command stream. A breakpoint is a flag you can insert anywhere in a macro that tells Tecplot to immediately suspend evaluation. Tecplot stops the action of a playing macro at the breakpoint to allow you to explore what is happening at that point in the macro file.

To add a breakpoint to a macro command:

1. Highlight the command in front of which you want to place the breakpoint by clicking on the command with the mouse cursor.
2. Click Add Break to add a breakpoint at the selected macro command. A **B** displayed at the beginning of the highlighted macro command indicates the breakpoint's placement.

To delete a breakpoint from a macro command:

1. Highlight the command for which you want to delete the breakpoint by clicking on the command with the mouse cursor.
2. Click Delete Break to delete the breakpoint.

To delete all the breakpoints set in a macro, click Delete All Breaks. This removes all the breakpoints within a macro.

### 28.3.5. Watching Variable Values while Debugging

Another debugging feature that the Macro Viewer provides is the ability to specify and view specific user defined, or system defined internal variables—that is, to specify watch variables. Use the Macro Variables dialog, shown in Figure 28-4, that is displayed after clicking Watch Variable to specify and view watch variables.

To specify a watch variable in a macro:

1. Click Watch Variables in the Macro Viewer dialog to bring up the Macro Variables dialog.
2. Type the name of the variable you want to watch in one of the User-Defined or Internal Variable text fields.

Leave this dialog open, off to one side, to watch the changing values in the Value column to the right of the variable name as the macro is playing.

The Macro Variables dialog also automatically displays any loop iteration values and command parameter stack calls that occur as a macro is played. In the Loops display area, the Value column for loops displays the loop iteration counter and changes as the system cycles through the loop sequence. The End Value displays the total number of iterations set for that loop. Tecplot lists the iteration values for up to three levels of nested loops. The Call Stack area

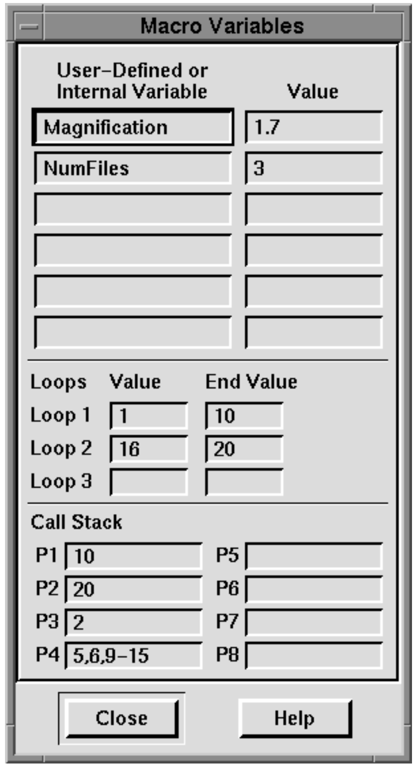



**Figure 28-4.** The Macro Variables dialog.

displays the parameter values used in calling the currently active macro function. Each field displays one parameter value up to the limit of eight parameters per macro function call. The display changes to show the state of the currently called macro function as Tecplot evaluates a **`$!RUNMACROFUNCTION`** command.

## 28.3.6. Modifying Macro Variables

After a breakpoint stops macro evaluation, you can change the value of a variable, a loop, or a stack call parameter. The ability to change these values in the middle of evaluating a macro allows you to either check a certain condition or to get out of a problem situation.

To change the value of a variable, a loop or a call stack parameter:

1. Highlight the variable value you want to change by selecting the value in the text field.
2. Type the new value.

3. Click Go or Step to continue the macro evaluation with the new value.

## 28.4. Doing More with Macros

There is much more to macro files than just recording and playing them. Tecplot provides you a whole macro programming language that allows you to develop tools for performing various functions, from developing your own interactive demonstration to transforming your data with a series of algorithms. See the *Tecplot Reference Manual* for details of Tecplot's macro language and macro files.

### 28.4.1. Processing Multiple Files

Suppose you need to create hardcopy plots with a specific style for a large number of input data files. You can do this with a macro that does the following:

- Reads in data files named `t`*nnn*`.plt` where *nnn* counts from 1 to 50.
- Applies a predefined layout to the data set.
- Generates a PostScript file named `t`*nnn*`.ps` (*nnn* counting from 1 to 50), one for each data file.

There are a number of ways to create a macro like the one described. The easiest way involves using the **`$!OPENLAYOUT`** command feature that allows you to replace data files referenced in the layout file itself with other data files:

1. Read in a representative example data file and create the layout you want to plot. If you are creating a contour plot make sure the contour levels you choose will be good ones for all files that are processed.
2. Create a macro file that references the layout file created in Step 1.

If the layout file generated in Step 1 is called **`cont.lay`** then the final macro to process the data files is as follows:

```
#!MC 900
#       Use a variable to store the number of
#       files to process.
$!VarSet |NumFiles| = 50
#       Make sure the output is PostScript.
$!ExportSetup
  ExportFormat    = PS
  Palette         = MONOCHROME
 #       Begin the loop
$!Loop |NumFiles|
#       Here is where we make use of the special feature
```

```
#       (i.e. the AltDataLoadInstructions option) of the $!OPENLAYOUT
#       command that allows us to override the named
#       data files within the layout file. Also make use of
#       the intrinsic LOOP macro variable.
$!OpenLayout "cont.lay"
  AltDataLoadInstructions = "t|LOOP|.plt"
#
#       Set the name of the file to be printed.
#
$!ExportSetup
  ExportFName = "t|LOOP|.ps"
#
#       Create the PostScript file.
#
$!Export
$!EndLoop
$!Quit
```

## 28.5. When to use Macros, Layouts or Stylesheets

Tecplot layout files are simply macro files. Why not just create the plot you want, then save a layout to preserve it, rather than recording a macro to recreate it? Layout files are typically very simple macro files; they do not include loops or data alterations or any such “programming.” Macros are generally used to do more complex tasks than layout files.

Tecplot processes layout and stylesheet files in a slightly different manner than it does macro files. In general, commands processed from a macro file undergo rigorous error checking and adjustments are sometimes made when values are outside certain limits. Layout and stylesheet files are treated more as a single unit by Tecplot. Individual commands are given more latitude because Tecplot assumes that the layout or stylesheet at one time represented a valid state. This is also why, while stepping through a macro file, the **$!OPENLAYOUT** and **$!READSTYLESHEET** commands are processed in a single step and you are not able to step “into” either of these commands. For layout and stylesheet files, error checking is not done until after the file is processed. By contrast, macro files are checked and adjusted line by line during processing. So, results from a layout file or stylesheet that you modify by hand may not be the same as those from a macro file.

# CHAPTER 29 Batch Processing

You can run Tecplot in batch mode to create plots without displaying any graphics to the screen. This saves a lot of time when processing multiple files for printing or export. In batch mode, Tecplot can be executed locally on your workstation computer or, under UNIX, remotely using an ASCII terminal. The only limitation for batch mode operation is that you cannot create export files in bitmap formats, since these files are generated from the screen.

## 29.1. Batch Processing Setup

To prepare for batch processing, you generally perform the following steps:

1. Create a macro file to control the batch processing. You may do this either interactively, by recording a Tecplot session, or using an ASCII text editor, or both. See Chapter 28, "Using Macros."
2. Create layout and stylesheet files as necessary.
3. Prepare data files.
4. Debug the macro file by running Tecplot while not in batch mode.

Macros are necessary to do batch processing. When Tecplot is launched in batch mode it requires that you provide the name of a macro file to execute. The minimal command to launch Tecplot in batch mode is as follows:

**`tecplot -b -p`** *macrofile*

The **`-b`** flag instructs Tecplot to run in batch mode and the **`-p`** *macrofile* tells Tecplot the name of the macro file to execute. See Appendix A, "Tecplot Command Line Options," for more command line options.

How the macro file interacts with layout and/or stylesheet files is the subject of the following sections. Different strategies work with different situations depending on your data and what you want to do with it.

## 29.2. Batch Processing Using a Layout File

Combining layout files with batch processing is both powerful and flexible. With layout files you can organize a plot using one or more frames in a single file. The layout file manages data sets and can be altered on the fly, either on the command line or within a macro that loads the layout file.

For example, suppose you want to do the following sequence of tasks in batch mode:

- Load a data file from a user supplied file name.
- Create a specific style of plot.
- Create a PostScript file of the plot.

You can set up the batch job as follows:

1. Obtain a representative data file to be plotted.
2. Interactively, create a layout of the style of plot you want. (For this example, name the file **`batch.lay`**).
3. Create (using a text editor) the following macro (for this example call this macro **`batch.mcr`**):

```
#!MC 900
$!ExportSetup
  ExportFormat = PS
  Palette = MONOCHROME
$!Export
$!Quit
```

The above macro can be modified to choose a different driver or palette depending on your situation.

Use the following command to run the job in batch mode:

**`tecplot -b -p batch.mcr -y tecplot.out batch.lay`** *mydatafile*

Layout files are self-contained. They contain all the information necessary to create a plot including the name(s) of the data file(s) to load. When you supply the names of data files on the command line (*mydatafile* in the above example) along with the layout file, Tecplot replaces the data files referenced in the layout file with the ones from the command line. If two or more data files are to be combined to form a single data set, use the “**+**” symbol to join the data file names.

The final result is a file called **`tecplot.out`** which contains the PostScript commands.

# 29.3. Processing Multiple Data Files

In Section 29.2, “Batch Processing Using a Layout File,” we set up Tecplot to process a user-supplied data file (or data files) and create a single output file. If the above procedure is to be repeated for a large number of input files (one at a time), you can do this by using a loop: either outside Tecplot in the operating system or within Tecplot using the flow-of-control commands in the Tecplot macro language.

## 29.3.1. Looping Outside Tecplot

The following examples show the command files for launching Tecplot in an operating system loop on two different operating systems. Tecplot processes five data files named **d***nn***.plt** and creates ten output files named **d***nn***.out** where *nn* goes from 1 to 10.

**29.3.1.1. Looping Outside Tecplot (UNIX).** Create a shell script with the following commands:

```
#!/bin/sh
n=1
while test $n -le 10
do
    tecplot -b -p batch.mcr -y d$n.out batch.lay d$n.plt
    n=‘expr $n+1‘
done
```

**29.3.1.2. Looping Outside Tecplot (Windows).** Create a batch file with the following commands:

```
for %%f in (d1 d2 d3 d4 d5 d6 d7 d8 d9 d10)
do tecplot -b -p batch.mcr -y %%f.out batch.lay %%f.plt
```

## 29.3.2. Looping Inside Tecplot

In Section 29.3.1, “Looping Outside Tecplot,” we set up Tecplot to process multiple data files using the operating system language to do the looping. There are two drawbacks to this procedure:

- The operating system languages are not portable between different operating systems.
- Tecplot must be continuously started and stopped each time a new data set is processed.

A more efficient approach is to loop through the data files inside Tecplot. Here, the layout file and the data files are all named within the Tecplot macro. The command line in this example is simple, as follows:

```
tecplot -b -p batch.mcr
```

The Tecplot macro is set up as follows:

```
#!MC 900
$!EXPORTSETUP
  EXPORTFORMAT = PS
  PALETTE = MONOCHROME
$!LOOP 10
$!OPENLAYOUT "batch.lay"
  ALTDATALOADINSTRUCTIONS = "d|LOOP|.plt"
$!EXPORTSETUP
  EXPORTFNAME = "d|LOOP|.out"
$!EXPORT
$!ENDLOOP
$!QUIT
```

The **$!OPENLAYOUT** command loads in **batch.lay** but replaces the data file referenced in the layout with the file names in the **ALTDATALOADINSTRUCTIONS** sub-command. The **$!EXPORTSETUP** command is used in two places. Initially it is used to set the export format. Later it is used just to change the name of the file to export to. The **$!EXPORT** command does the actual exporting.

## 29.4. Batch Processing Using Stylesheet Files

Instead of using layout files, you can use stylesheet files when batch processing. In general, batch processing with stylesheets works just like the batch processing described above in “Processing Multiple Data Files,” except a stylesheet file is used instead of a layout file.

If you want to make many different plots using the same data set, stylesheets will be more efficient than layout files.

## 29.5. Batch Processing Diagnostics

Each time Tecplot is run in batch mode it creates a file defined by the name in the **BATCHLOGFILE** environment variable, or, if the environment variable is not defined, by a file named **batch.log** in the directory where Tecplot was started. If the name given in the **BATCHLOGFILE** environment variable is a relative path, the directory name where Tecplot was started is prepended. A running commentary on actions performed in Tecplot, as well as warning and error messages, are sent to the **batch.log** file.

## 29.6. Moving Macros to Different Computers or Different Directories

The file **`tecplot.phy`** is created each time you run Tecplot interactively. It contains information about the physical characteristics of your computer system as well as information about the size of the Tecplot process window used during the last Tecplot session. It also contains the name of the last layout file used by Tecplot. If you are developing macros on one computer, but using them for batch processing on a different computer, you must transfer the **`tecplot.phy`** file from the development computer to the computer where you will run Tecplot in batch mode. Under UNIX, the same is true if you are developing macros in one directory, but will be processing them in batch mode in a different directory. See Section 31.6, “Configuring the Location of the "tecplot.phy" File,” for information on the location of your **`tecplot.phy`** file.

# CHAPTER 30 Animation and Movies

Animation is the process of showing a succession of still images to give the impression of motion. Tecplot provides a variety of methods for creating animated plots, and exporting them to movie files for playback at a later time. There are three basic animation methods available:

- **Animation Tools:** Perform simple animations using the dialogs in the Animate sub-menu under the Tools menu. The Animate sub-menu allows you to animate zones, XY- mappings, contour levels, IJK-planes, IJK-blanking, and streamtraces. The animation is viewed within Tecplot, or exported to a movie file which can be played back outside of Tecplot.
- **Manually:** Interactively create movies by creating an initial plot, exporting the image as either a AVI or Raster Metafile movie, then repeatedly changing and appending new images to the same movie file.
- **Macros:** Use a macro to perform multiple, repetitive changes, and write each image to a movie file.

This chapter discusses these methods, as well as ways to play back movies that have been created in Tecplot.

## 30.1. Animation Tools

Use the Animate sub-menu under the Tools menu to have Tecplot cycle through your data, automatically displaying zones, IJK-planes, or any of several other plot elements, one after the other, until your entire data set has been displayed. The following plot elements may be animated using the dialogs in the Animate submenu:

- **Zones.**
- **XY-Mappings.**
- **Contour Levels.**

- **IJK-Planes.**
- **IJK-Blanking.**
- **Slices.**
- **Streamtraces.**

### 30.1.1. Animating Zones

Use the Animate Zones dialog to display all or a specified subset of the zones in the current data set, one at a time.

To animate zones:

1. From the Tools menu, choose Animate, then choose Zones. The Animate Zones dialog appears as shown in Figure 30-1.

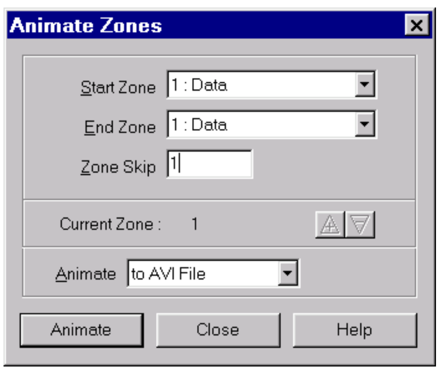



**Figure 30-1.** The Animate Zones dialog.

2. Specify a start zone (the first zone you want displayed), an end zone, and a zone skip in the fields provided. If you specify a start zone having a higher number than the end zone, Tecplot cycles backward from the start to the end.
3. If you want Tecplot to create a movie file containing the animation, select “to AVI file” or “to RM file” from the Animate drop-down menu.
4. Click Animate to run the animation automatically, or use + and - in the Current Zone area to “step through” the animation one zone at a time. (These cycle through the range of zones specified by Start Zone and End Zone; if your range is reversed, so are their actions.)

For example, suppose you have several sets of 2-D data defined at different times. At each time value, the data point positions are the same, but the variables defined at the data points are different. You could organize this body of data into one data file where each time value is allocated to a separate zone. In Tecplot, you could set up the plot style the way you want for all zones, then use Animate Zones to view the data for all time values by activating each zone, one at a time.

As an example, you might try loading the data file **`demo/plt/cylinder.plt`** and animating its zones. The cylinder data set has three zones.

### 30.1.2. Animating XY-Mappings

Use the Animate XY-Mappings dialog to display all or a specified subset of the XY-mappings defined in the current frame, one at a time.

To animate XY-mappings:

1. From the Tools menu, choose Animate, then choose XY-Mappings. The Animate XY Mappings dialog appears, as shown in Figure 30-2.

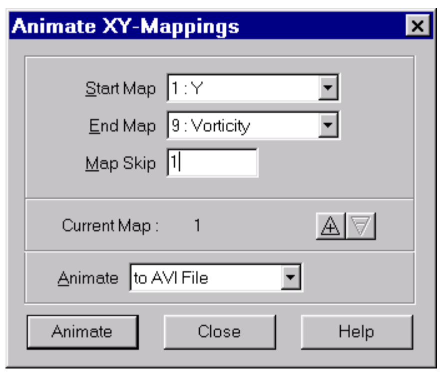



**Figure 30-2.** The Animate XY-Mappings dialog.

2. Specify a Start Map (the first XY-map you want displayed), an End Map, and a Map Skip in the fields provided. If you specify a Start Map having a higher number than the End Map, Tecplot cycles backward from the start to the end.
3. If you want Tecplot to create a movie file containing the animation, select “to AVI file” or “to RM file” from the Animate drop-down menu.

4. Click Animate to run the animation automatically, or use + and - in the Current Map area to "step through" the animation one XY-map at a time. (Theses cycle through the range of XY-maps specified by Start Map and End Map; if your range is reversed, so are their actions.)

You can try this with the demo data file, **`tec90/demo/plt/rain.plt`**.

### 30.1.3. Animating Contour Levels

Use the Animate Contour Levels dialog to display all or a specified subset of the contour levels defined in the current frame, one at a time.

To animate contour levels:

1. From the Tools menu, choose Animate, then choose Contour Levels. The Animate Contour Levels dialog appears, as shown in Figure 30-3.

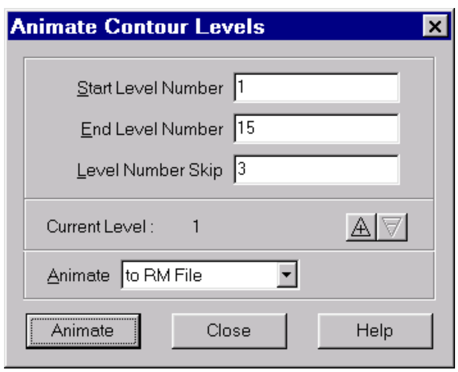



**Figure 30-3.** The Animate Contour Levels dialog.

2. Specify a start contour level (the first contour level you want displayed), an end contour level, and a contour level skip in the fields provided. If you specify a start level having a higher number than the end level, Tecplot cycles backward from the start to the end.
3. If you want Tecplot to create a movie file containing the animation, select "to AVI file" or "to RM file" from the Animate drop-down menu.
4. Click Animate to run the animation automatically, or use the + and - in the Current Level area to "step through" the animation one contour level at a time. (These cycle through the range of levels specified by Start Level Number and End Level Number; if your range is reversed, so are their actions.)

### 30.1.4. Animating IJK-Planes

Use the Animate IJK-Planes dialog to display all or a specified sub-set of the IJK-planes in the current data set, one at a time. You can choose to animate either the I-, J-, or K-planes.

To animate IJK-planes:

1. From the Tools menu, choose Animate, then choose IJK-Planes. The Animate IJK-Planes dialog appears, as shown in Figure 30-4.

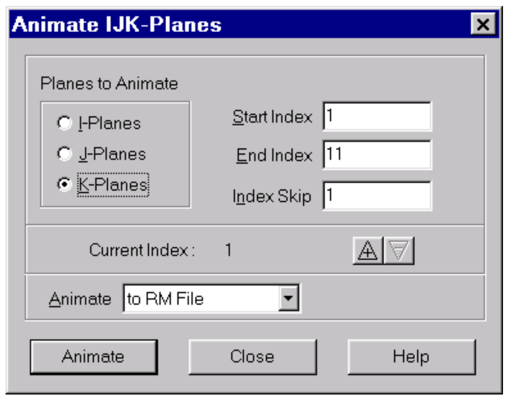



**Figure 30-4.** The Animate IJK-Planes dialog.

2. Specify the set of planes to animate: I-Planes, J-Planes, or K-Planes.
3. Specify a start index (the first plane you want displayed), an end index, and an index skip in the fields provided. If you specify a start index having a higher number than the end index, Tecplot cycles backward from the start to the end.
4. If you want Tecplot to create a movie file containing the animation, select “to AVI file” or “to RM file” from the Animate drop-down menu.
5. Click Animate to run the animation automatically, or use + and - in the Current Index area to “step through” the animation one plane at a time. (These cycle through the range of planes specified by Start Index and End Index; if your range is reversed, so are their actions.)

Figure 30-5 shows an example of animating I-planes in an IJK-ordered zone.

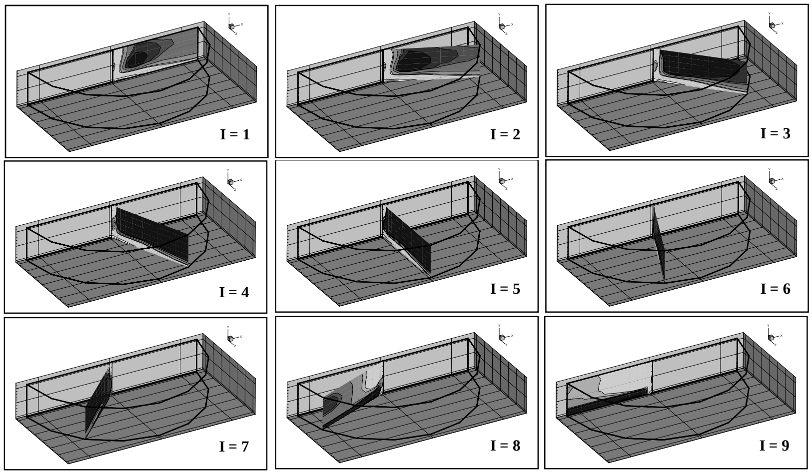



**Figure 30-5.** An animated sequence of I-planes.

### 30.1.5. Animating IJK-Blanking

Use the Animate IJK-Blanking dialog to animate a sequence of Tecplot renderings starting with an initial set of blanked IJK indices and proceeding in a series of interpolated steps to a final set of blanked IJK indices. Before you can animate IJK-blanking, you must first specify which zone you want to use. Do this in the IJK-Blanking dialog.

To animate a sequence of IJK-blankings, you must first turn on IJK-blanking, then use the Animate IJK-Blanking dialog:

1. From the Style menu, choose IJK-Blanking. The IJK-Blanking dialog appears.
2. Select the Include IJK-Blanking check box. The remaining controls in the dialog become active.
3. From the Zone drop-down, select the desired zone and specify whether the interior or exterior of the zone should be blanked.
4. Click Close.
5. From the Tools menu, choose Animate, then choose IJK-Blanking. The Animate IJK-Blanking dialog appears, as shown in Figure 30-6.
6. Specify an initial set of blanked IJK-indices in the text fields grouped under the title Starting Index (% of Max). Enter a range of indices for each of I, J, and K. Index values are entered as percentages of the maximum index.

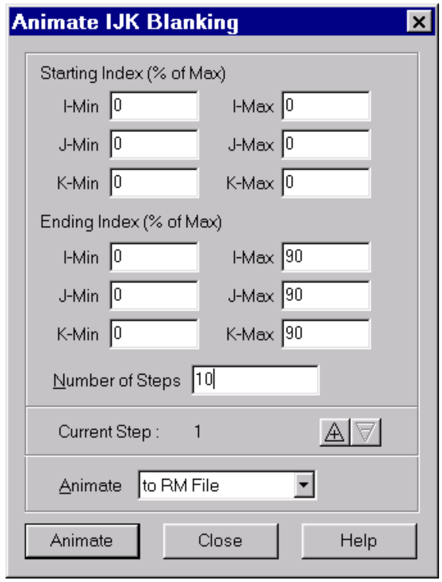



**Figure 30-6.** The Animate IJK-Blanking dialog.

7. Specify a final set of blanked IJK-indices in the text fields grouped under the title Ending Index (% of Max). Enter a range of indices for each of I, J, and K.
8. Specify the number of steps—that is, the number of renderings required to move from the initial IJK-blanking to the final IJK-blanking. The minimum number of steps is two.
9. If you want Tecplot to create a movie file containing the animation, select “to AVI file” or “to RM file” from the Animate drop-down menu.
10. Click Animate to run the animation automatically, or use + and - in the Current Step area to “step through” the animation one step at a time.

### 30.1.6. Animating Slices

Use the Animate Slices dialog to animate a sequence of slices through your data. Use the Slice tool or the 3D Slice Details dialog to configure the start and end slices for your animation. The Animate Slices dialog is shown in Figure 30-7.

To animate a sequence of slices perform the following steps:

1. Use the Slice tool or the 3D Slice Details dialog to define a start and end slice.
2. From the Tools menu, choose Animate, then select Slices. The Animate Slices dialog appears, as shown in Figure 30-7.

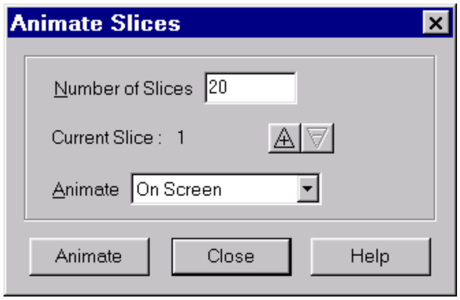



**Figure 30-7.** The Animate Slices dialog.

3. Enter the number of slices to animate.
4. If you want to create a movie file containing the animation select “to AVI file” or “to RM file” from the Animate drop-down menu.
5. Click Animate.

If the slices are currently assigned to I-, J-, or K-planes, the total number of slices you can animate is limited to the total number of I-, J-, or K-planes possible between the start and end slice planes. If the slice planes are X-, Y-, or Z-planes you may specify any number of slices larger than or equal to two.

## 30.1.7. Animating Streamtraces

Use the Animate Streamtraces dialog to create animated images of streamtraces. You specify the number of images shown for each streamtrace cycle, and the number of cycles to show. The resulting animation shows streamtrace markers and/or streamtrace timing dashes at each step of the animation, “moving” down the streamtrace. Before you can animate streamtraces, you must turn on either the timing dashes or timing markers or both, using the Streamtrace Details dialog under the Field menu. See Section 13.6, “Streamtrace Timing,” for details.

To animate your streamtraces:

1. Create a number of streamtraces. (For information on how to do this, see Chapter 12, “Streamtraces.” )
2. If you have not already done so, select Show Dashes or Show Markers from the Timing page of the Streamtrace Details dialog. (Choose Streamtrace Details from the Field menu to call up this dialog.)
3. From the Tool menu, choose Animate, then choose Streamtraces. The Animate Streamtraces dialog appears, as shown in Figure 30-8.

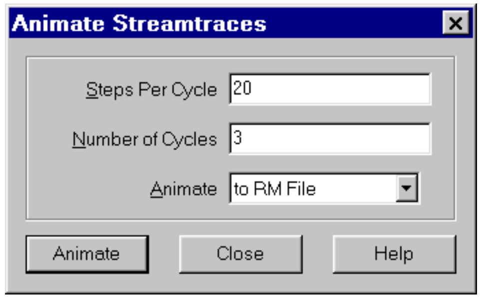



**Figure 30-8.** The Animate Streamtraces dialog.

4. If you want Tecplot to create a movie file containing the animation, select "to AVI file" or "to RM file" from the Animate drop-down menu.
5. Specify the number of steps per cycle and the number of cycles in the fields provided.
6. Click Animate to begin the animation.

### 30.1.8. Creating a Movie File

Each of the animation dialogs offers you the option of saving the current animation as an AVI or Raster Metafile (Framer Movie File).

To save the current animation as a movie file:

1. In the appropriate Animation dialog, select "to AVI file" or "to RM file" from the Animate drop-down menu. When you click Animate, the appropriate Export dialog will appear.
2. Choose the image width and whether to use multiple color tables.
3. If you choose AVI as the export file type, you may specify animation speed in frames per second. Animation speed is only available for AVI files.
4. When you click OK, the Select Move File dialog will appear. Enter a file name.
5. Click OK to create the movie file. The images are written to the movie file while the Working dialog is displayed on your screen. (The screen image will not change.)

See Section 30.5.2, "Viewing Raster Metafiles in Framer," for information on running Framer on your movie files.

## 30.2. Creating a Movie Manually

You can create a sequence of Raster Metafile images interactively as follows:

1. Create the first plot.
2. Select Export from the File menu and select Raster Metafile from the list.
3. Setup the desired characteristics on the Raster Metafile dialog, and click OK.
4. Select a file name when the Select Export File dialog is presented, then click Save.
5. The first image will be saved after you click Save on the Select Export File dialog, then the Record Animation File dialog will appear.
6. Create the next plot interactively.
7. When satisfied with the next image, click Record Next Image on the Record Animation File dialog.
8. Repeat steps 6 and 7 to continue adding images to the animation. When done, click Finish Animation on the Record Next Image dialog.

## 30.3. Creating Movies with Macros

The Tecplot macro language expands the capabilities of Tecplot's standard animation features. The macro commands allow you to do almost anything you can do interactively, and export images to movie files. You can also use loops to repeatedly rotate 3-D objects, cycle from one active zone to another, and so on, to create your movie. See Chapter 28, "Using Macros," for detailed information regarding the Tecplot macro language.

A typical macro file for making movies has the following form:

```
#!MC 900
 ... optional commands to set up the first image
$!EXPORTSETUP
  EXPORTFORMAT = AVI
  EXPORTFNAME = "mymovie.avi"
$!EXPORTSTART
$!LOOP 50
 ... commands to set up next image
  $!REDRAWALL
  $!EXPORTNEXTFRAME
$!ENDLOOP
$!EXPORTFINISH
```

For example, the following macro file duplicates the actions performed by the Animate Zones dialog:

```
#!MC 900
##  Set up Export file type and file name.
$!EXPORTSETUP
```

```
  EXPORTFORMAT = AVI
  EXPORTFNAME = "C:\temp\timeseries.avi"
##  Begin Animating
$!LOOP |NUMZONES|
## Change the active zone.
## The |Loop| variable is equal to the current
## loop cycle number.
  $!ACTIVEFIELDZONES = [|Loop|]
  $!REDRAWALL
##  This series of $!IF statements ensures
##  that a new AVI file will be created when
##  the macro is started.
  $!IF |Loop| == 1
    $!EXPORTSTART
  $!ENDIF
  $!IF |Loop| != 1
    $!EXPORTNEXTFRAME
  $!ENDIF
$!ENDLOOP
$!EXPORTFINISH
```

To run this macro, do the following:

1. Create a macro file like the example above. Edit the macro file to specify a new export file name if necessary.
2. Load in a multi-zone data file.
3. Switch to 2D frame mode.
4. From the File menu, choose the Play option from the Macro sub-menu, then choose the macro file you specified in step 1.

**Note:** Version 8.0 macros for creating animation files (Raster Metafiles or AVI) must be revised to work with Version 9.0, as shown in the example above.

# 30.4. Advanced Animation Techniques

There may be times when you want to include information in your animation which tells viewers about the time step, current zones, or an XY-map. There are several ways this can be done.

## 30.4.1. Changing Image Size of Animations

When you need a particular size for your animation image, such as 300 by 250 pixels, first edit your frame to the correct width and height. Then export only the current frame.

### 30.4.2. Changing Text in Animations by Attaching Text to Zones

This method works best if you are using the Zone Animation function under the Tools menu. First, create several text strings in your data file, and use the **ZN=** parameter to attach each text string to a zone or XY-mapping. (See section 5.1.3, "Text Record," for details on attaching text to zones.) You should have a separate text string for each zone in your data set that will be used in your animation. An example of this is:

```
ZONE T="Temp. distribution, Time = 0.5 seconds" I=51, J=51 F=POINT
.
.
.
list of variable values
.
.
.
TEXT X=70, Y=90, T="Time = 0.5 seconds", F=COURIER, CS=FRAME, H=2, ZN=1

ZONE T="Temp. distribution, Time = 1.0 seconds" I=51, J=51 F=POINT
.
.
.
list of variable values
.
.
.
TEXT X=70, Y=90, T="Time = 1.0 seconds", F=COURIER, CS=FRAME, H=2, ZN=2
```

Next, use the Animate Zones or XY Maps tool to create your animation. Only the text string attached to the current zone in your animation will be visible. You can also use Tecplot's dynamic text feature (see Section 18.1.7. "Adding Dynamic Text") to insert a zone name into your text strings. For example:

```
ZONE T="Time = 1.0 seconds" I=51, J=51 F=POINT
.
.
.
list of variable values
.
.
.
TEXT X=70, Y=90, T="&(ZONENAME:2)", F=COURIER, CS=FRAME, H=2, ZN=2
```

### 30.4.3. Changing Text in Animations by Using the Scatter Symbol Legend

You also can show the name of the current zone in your animation with the use of the scatter legend. Although you may not be using Scatter zone layer in your plot, the scatter legend will show the name of the current zone during the animation. See Section 14.7, "Creating a Scatter Legend," for more information about using the scatter legend in Tecplot.

Turn on the scatter legend by selecting the Scatter Legend option on the Field menu, then make the scatter symbol invisible in the legend by changing the scatter color to white for all of your zones in the Scatter Attributes dialog (or changing to the same color as your frame background if it is not white).

### 30.4.4. Changing Text in Animations by Using Macros

If you are using a macro to generate your animation, you can include a command to attach a text string that contains the current time step:

```
$!Loop 20
  $!Pick AddAll
    SelectText = TRUE
  $!Pick Clear
  .
  .
  .
  Commands to change the existing plot
  .
  .
  .
  $!AttachText
    Text = "Time = |Elapsed_Time| seconds."
    XYPos
    {
     X = 70
     Y = 90
    }
  $!Redraw
 $!If |Loop| == 1
  $!ExportStart
  $!EndIf
  $!If |Loop| == 1
  $!ExportNextFrame
  $!EndIf
    $!Varset |Elapsed_Time| += 0.5
$!EndLoop
```

The above loop is an example of how you could use a user-defined macro variable (discussed in the *Tecplot Reference Manual*) to insert an increasing time value into a text string. This example uses a variable called **|Elapsed_Time|**, which we consider to be the current time. One could alternatively set **|Elapsed_Time|** to be a text string that uses dynamic text:

```
$!Varset |Elapsed_Time| = "%(ZONENAME:|LOOP|)"
```

With each loop iteration, the **|LOOP|** macro variable will increment and the name for each zone will be inserted into **|Elapsed_Time|** each time through the loop. (Zone 1, 2, 3, 4, and so on.)

### 30.4.5. Animating Multiple Frames Simultaneously

Animation of plots in multiple frames requires the use of a macro. The **$!FRAMECONTROL PUSHTOP** command is used to switch between each frame. The following template demonstrates how this is done with a layout where each frame contains a similar plot:

```
#!MC 900

##Set the number of images (movie frames) in the animation.
$!VARSET |NumCycles| = 10

$!EXPORTSETUP
  EXPORTFNAME   = "2frames.rm"
  EXPORTFORMAT  = RASTERMETAFILE
  BITDUMPREGION = ALLFRAMES
.
.
Insert commands to set up first frame, if necessary.
.
.
## Outer loop.
$!LOOP |NumCycles|
  ##  Inner loop cycles through each frame in the current layout.
  $!LOOP |NumFrames|
    .
    .
    Insert commands to change the plot in the current frame.
    .
    .
    ##This command pushes the topmost (active) frame to the back,
    ##making the next frame active.
    $!FrameControl PushTop
  $!EndLoop
  ##  This series of $!IF statements ensures
  ##  that a new AVI file will be created when
  ##  the macro is started.
  $!IF |Loop| == 1
    $!EXPORTSTART
  $!ENDIF
  $!IF |Loop| != 1
    $!EXPORTNEXTFRAME
```

```
  $!ENDIF
$!ENDLOOP

$!EXPORTFINISH
```

# 30.5. Viewing Movie Files

The following tools allow you to view movie files you have created with Tecplot.

## 30.5.1. Viewing AVI Files

AVI format is the standard video format for Windows platforms. Below are some applications that can be used to view and/or edit AVI files:

- **Media Player:** A standard movie viewer included with Windows.
- **Xanim:** A program for playing a wide variety of video formats on UNIX X11 machines. More information is available at xanim.va.pubnix.com.
- **Premier:** A powerful tool for professional digital video editing. More information is available at www.adobe.com.

## 30.5.2. Viewing Raster Metafiles in Framer

Raster Metafile is a NASA-defined standard format for storing bit images and may contain one or more images. You can create a Raster Metafile in Tecplot either interactively, or using a Tecplot macro. For many types of repetitive plots (such as rotations, where each image is a slightly rotated version of the previous image), macros provide a very convenient means of simplifying Raster Metafile creation.

The Raster Metafile format is defined in the following reference:

> Taylor, N., Everton, E., Randall, D., Gates, R., and Skeens, K., NASA TM 102588, *Raster Metafile and Raster Metafile Translator.* Central Scientific Computing Complex Document G-14, NASA Langley Research Center, Hampton, VA. September, 1989.

Once you have created your Raster Metafile, you can view the resulting file with Framer. Framer is a utility program that is included with Tecplot. It allows you to view files stored in Raster Metafile format and runs independently of Tecplot. You may freely distribute Framer so that others may view your movies.

The Motif version of Framer is run from your shell prompt; the Windows version can be launched from the Tecplot program folder under the Start button (Windows 95 or 98, or Windows NT 4.0). You may freely distribute the Framer executable to allow others to view your animation.

To launch Framer at a command line (shell prompt, Run command, and so forth), use the following command:

**framer** *[options] [rmfile]*

where *[rmfile]* is the name of a file containing Raster Metafile bitmaps created by Tecplot, and *[options]* is one or more of the options listed in Section 1 of Appendix B, "Utility Command Line Options."

To run Framer on UNIX type:

**framer** *[filename]*

If you do not specify a file name, Framer prompts you for one. (Under Windows you get the file dialog shown in Figure 30-9. In this dialog, you can choose to set buffering [equivalent to the **-b** flag] and/or multiple color maps [equivalent to the **-m** flag].) For a list of Framer

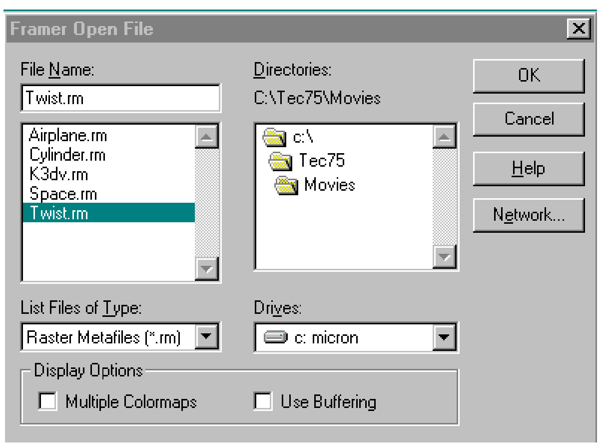



**Figure 30-9.** The Framer Open File dialog under Windows.

command lines, see Appendix B.1, "Framer."

Figure 30-10 shows the main Framer window under Windows.

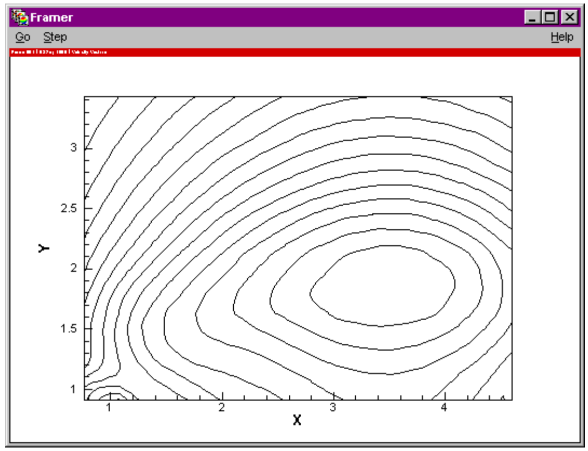



**Figure 30-10.** The Framer application window under Windows.

# CHAPTER 31 Customizing Tecplot

Tecplot comes with a complete set of factory defaults for creating plots of all kinds: frame attributes, such as the initial frame size, background color; axis attributes, such as axis line color and range; plot attributes, such as boundary line thickness and contour plot type; and so on. You can modify virtually all of these defaults through the use of a configuration file. A Tecplot configuration file is a special type of Tecplot macro file that Tecplot reads on start up; the settings in the configuration file override Tecplot's factory defaults.

You can create a configuration file from scratch, using any ASCII text editor, or you can have Tecplot create one for you using the Save Configuration option in the File menu's Preferences sub-menu. You can also use the Preferences sub-menu and the Display Performance dialog to modify many settings interactively.

This chapter discusses creating and editing configuration files. The names of the files used will vary from platform to platform; this chapter concentrates on UNIX and Windows files.

## 31.1. Tecplot Configuration Files

Tecplot looks for configuration files in one of three places: the current working directory, the user's home directory, and the Tecplot home directory. In the current working directory, Tecplot searches for a file named **`tecplot.cfg`**. If it finds it, it uses the settings in that file to override the factory defaults. If not, Tecplot searches your home directory for a file named **`.tecplot.cfg`** (**`tecplot.cfg`** in Windows). If Tecplot finds this file, it uses the settings in that file to override factory defaults. If there is no **`.tecplot.cfg`** file in your home directory, Tecplot looks in the Tecplot home directory for a file named **`tecplot.cfg`**, and uses the settings in that file to override the factory defaults. (Under Windows, your home directory is determined by the two environment variables **`HOMEDRIVE`** and **`HOMEPATH`**. If they are not set, Tecplot skips your home directory.)

You can save configuration files with any valid file name. To have Tecplot use them, you must either rename them to have one of the names Tecplot searches for by default, or, more com-

monly, use the "**-c**" command line option when starting Tecplot. For example, the following command starts Tecplot using the configuration settings in the file **mydefs.cfg**:

```
tecplot -c mydefs.cfg
```

System administrators can use the **tecplot.cfg** file in the Tecplot home directory to set system-wide defaults, then others on the system can copy the system configuration file to their own home directories and make any desired changes. The settings in your local configuration file are used instead of the settings in the system configuration file. A configuration file needs to include only those options for which you want to override defaults.

Tecplot under Motif has a second type of configuration file, an X11 resource file (app-defaults file) that controls the appearance of the Tecplot application and its dialogs. Most users do not need to concern themselves with this file; nothing in the resource file has any affect on the plots you create with Tecplot, either on screen or on paper. However, if you are an experienced Motif and X11 user, you may want to modify some of the resources to improve the appearance of Tecplot's windows and dialogs on your display. Section 31.4, "Configuring the Interface under UNIX," explains how to do this.

### 31.1.1. Creating a Configuration File

The simplest way to create a configuration file is to change the appropriate settings using the Tecplot interface, then save the configuration. For example, suppose you want to have your paper orientation default to portrait and have your default export format be Encapsulated PostScript (EPS). You can modify the settings using the appropriate Tecplot dialogs, then save the configuration file.

To save a Tecplot configuration file:

1. Change settings as desired using Tecplot dialogs.
2. From the File menu, choose Preferences, then choose Save Configuration. The Save Configuration dialog appears as shown in Figure 31-1.
3. Check the file name listed. If the file name is correct, click OK. If you want to save to a different file, click Change File Name and specify a new file name.


**Figure 31-1.** The Save Configuration dialog.

Here is the configuration file resulting from the changes described above:

```
#!MC 900
$!PAPER
  ORIENTPORTRAIT = YES
$!EXPORTSETUP
  EXPORTFORMAT = EPS
$!FRAMELAYOUT
   XYPOS
    {
    X = 1
    Y = 0.25
    }
  WIDTH = 9
  HEIGHT = 8
```

This configuration file specifies portrait orientation, and sets the Export format to EPS, just as desired. The only other settings saved are the default frame layout settings.

You can, however, obtain a configuration file that includes most of the factory defaults as follows:

1. From the File menu, choose Preferences, then choose Save Configuration. The Save Configuration dialog appears.
2. Select the check box labeled Include Factory Defaults.

3. Check the file name listed. If the file name is correct, click OK. If you want to save to a different file, click Change File Name and specify a new file name.

The created file contains factory defaults for the following types of Tecplot settings:

- Interface details.
- RGB color assignments for Tecplot's basic colors.
- Default paper layout.
- Print and export setup information.

If you modify any setting from these four types interactively and then save your configuration, the modifications are saved. However, modifications to other types of settings will not be saved.

You are not limited, however, to changing merely those settings which appear in the saved configuration file. Most settings which can be modified by one of Tecplot's SetValue macro commands can be changed in the configuration file. These other settings must be changed, however, by editing the configuration file.The simplest way to do this is to create a layout or macro with the settings you want, then copy and paste the appropriate SetValue commands into your configuration file. (Once you become more familiar with the macro language, it may be simpler to type in the appropriate SetValue command directly.) See the *Tecplot Reference Manual* for complete details on the SetValue and other macro commands.

For example, suppose you want your 2-D axes to appear cyan. You can add this preference to your configuration file as follows:

1. Using the Tecplot interface, create a 2-D plot with cyan axes, either recording your steps as a macro, or saving the result as a Tecplot layout.
2. Edit the resulting macro or layout, scanning for the lines that set the 2-D axis colors. The following example shows the commands that specify the X- and Y-axis details in a layout of a 2-D plot with cyan axes:

```
$!TWODAXIS
  XDETAIL
    {
    RANGEMIN = -2.99985003471
    RANGEMAX = 15.001799985
    GRSPACING = 5
    AXISCOLOR = CYAN
    }
  YDETAIL
    {
    RANGEMIN = -2.99985003471
    RANGEMAX = 13.4283224277
```

```
GRSPACING = 2
AXISCOLOR = CYAN
}
```

3. Discard everything but the lines that actually set the color:

```
$!TWODAXIS
  XDETAIL
    {
    AXISCOLOR = CYAN
    }
  YDETAIL
    {
    AXISCOLOR = CYAN
    }
```

4. Paste the resulting lines into your configuration file.

### 31.1.2. Setting Plot Defaults

A single **$!FIELD** command can be included to set plot defaults. The command cannot specify a zone, and is not effective for values set dynamically by Tecplot, such as Mesh Color. In the example below, the default contour type is Flood, scatter symbol shape is Delta, and scatter size is 1.8.

```
$!FIELD
   CONTOUR
     {
     CONTOURTYPE = FLOOD
     }
   SCATTER
     {
     FRAMESIZE = 1.8
     SYMBOLSHAPE
       {
       GEOMSHAPE = DEL
       }
     }
```

In the same way, a single **$!XYMAP** command can be added for XY-mapping defaults. In the example below, XY-mapping will have a dashed line pattern, and symbols will be filled circles.

```
$!XYMAP
   LINES
     {
     LINEPATTERN = DASHED
```

```
    }
  SYMBOLS
      {
      SYMBOLSHAPE
         {
         GEOMSHAPE = CIRCLE
         }
      ISFILLED = YES
      }
```

### 31.1.3. Configuring the Tecplot Interface

The following macro commands help you configure Tecplot's user interface and graphics drawing capabilities. They are all members of the **$!INTERFACE** macro. Although some of these commands can be executed in any Tecplot macro the best place to put these is in the Tecplot configuration file, **tecplot.cfg**.

#### 31.1.3.1. General Interface Configuration Options.

**$!INTERFACE MIDDLEMOUSEBUTTONMODE = (REDRAW, REVERTTOSELECT)**

Specify the action of the middle mouse button click. **REDRAW** redraws the current frame. **REVERTTOSELECT** forces the mouse tool back to the Selector tool. The default is **REDRAW**. This command can only be executed from the Tecplot configuration file.

**$!INTERFACE RIGHTMOUSEBUTTONMODE = (REDRAW, REVERTTOSELECT)**

Specify the action of the right mouse button click. **REDRAW** redraws the current frame. **REVERTTOSELECT** forces the mouse tool back to the Selector tool. The default is **REVERTTOSELECT**. This command can only be executed from the Tecplot configuration file.

**$!INTERFACE USETECPLOTPRINTDRIVERS = (YES, NO)**

Make Tecplot use its own internal print drivers instead of the Windows print drivers. The default for windows is **NO**. For UNIX this command has no effect. This command can only be executed from the Tecplot configuration file.

**$!INTERFACE SECURESPOOLCOMMANDS = (YES, NO)**

Specify whether or not spooler commands can only be altered from within of the tecplot configuration file. The default is **YES**. (You are only allowed to assign print spool commands from within the Tecplot configuration file itself.) This command can only be executed from the Tecplot configuration file.

**$!INTERFACE UNIXHELPBROWSERCMD =** *string*

Specify the command to execute to launch the browser on UNIX systems for viewing the Help files. This command can only be executed from the Tecplot configuration file.

**$!INTERFACE BEEPONFRAMEINTERRUPT = (YES, NO)**

Set this to **YES** if you want Tecplot to beep whenever a drawing is interrupted.

**$!INTERFACE PICKHANDLEWIDTH =** *d*

Set the width of pick handles. *D* is in units of inches as measured on the screen.

**$!INTERFACE RULERTHICKNESS =** *d*

Set the thickness (width) of the rulers in the work area. *D* is in units of inches as measured on the screen.

**$!INTERFACE RULERPADDING =** *d*

Set the distance between the ruler and the part of the work area usable for drawing in frames.

**$!INTERFACE SHOWCOORDINATES = (YES, NO)**

Show or do not show the running coordinates in the status line.

**$!INTERFACE SHOWSTATUSLINE = (YES, NO)**

Show or do not show the entire status line.

**$!INTERFACE SHOWCONTINUOUSSTATUS = (YES, NO)**

Show or do not show the continuous status messages in the status line.

**$!INTERFACE SHOWWAITDIALOGS = (YES, NO)**

You can disable the launch and display of all Wait dialogs by setting this to **NO**. (Wait dialogs are launched during long operations and give you the ability to cancel the operation.)

**$!INTERFACE SHOWFRAMEBORDERSWHENOFF = (YES, NO)**

If set to **NO** frame borders are drawn on the screen using a dashed line when you elect to turn them off (from the Frame/Edit dialog). If set to **YES** they are not drawn at all on the screen. This setting only applies to the screen and does not effect print output or image output.

**$!INTERFACE PRINTDEBUG = (YES, NO)**

Set this to **YES** to get a running commentary of your Tecplot session written to the standard out (UNIX only).

**$!INTERFACE ENABLEINTERRUPTS = (YES, NO)**

If set to **NO** this will not allow you to interrupt drawing in the work area.

**$!INTERFACE ENABLEWARNINGS = (YES, NO)**

If set to NO this will disable the launch and display of all warning dialogs.

**$!INTERFACE ENABLEPAUSES = (YES, NO)**

If set to **NO** this will disable the launch of all Pause dialogs during the execution of macros.

**$!INTERFACE ENABLEDELAYS = (YES, NO)**

If set to **NO** all **$!DELAY** macro commands will be ignored during macro processing.

**$!INTERFACE ALLOWDATAPOINTSELECT = (YES, NO)**

If set to **NO** this disables the feature where you can use the Adjustor tool to move data points around on the screen for XY- and 2-D plots.

**$!INTERFACE USESTROKEFONTSONSCREEN = (YES, NO)**

If set to **YES** all text drawn in the work area will be drawn using Tecplot's internal stroke fonts. If set to **NO** the native True Type fonts will be used instead. This option has no effect under UNIX.

**$!INTERFACE USESTROKEFONTSFOR3DTEXT = (YES, NO)**

If set to **YES** all 3-D text drawn in the work area will be drawn using Tecplot's internal stroke fonts. 3-D text consists of ASCII scatter symbols, and node and cell labels when the current frame mode is 3-D. For 3-D text, this setting overrides the setting of **USESTROKEFONTSONSCREEN**. If set to **NO** the native True Type fonts will be used instead. This option has no effect under UNIX.

**$!INTERFACE MAXTRACELINES =** *n*

Set the approximate number of lines to draw when the data in a frame is traced.

**$!INTERFACE FASTLINEPATTERNSONSCREEN = (YES, NO)**

Set to **YES** (the default) to draw patterned lines on the screen using a fast approximate method. If set to **NO** all patterned lines on the screen will be drawn by hand.

**$!INTERFACE USEFASTAPPROXCONTINUOUSFLOOD = (YES, NO)**

Set to **YES** to use a faster but less accurate method for drawing continuous flooding in 3-D.

**$!INTERFACE FORCEGOURAUDFORCONT3DFLOOD = (YES, NO)**

If set to **YES** and the surface effects are paneled and the plot contains continuous flooding then the surface effects will be upgraded to Gouraud. Gouraud shading is much more efficient than paneled shading in this instance.

**$!INTERFACE USEDISPLAYLISTS = (YES, NO)**

Set to **YES** to cache graphics primitives into display lists. This provides much faster rendering speed but requires more memory.

**$!INTERFACE USEDOUBLEPRECISIONFORDISPLAYLISTS = (YES, NO)**

Set to **YES** to request that the graphics sub-system use double precision for vertex points and the depth buffer. This is only a request that the graphics sub-system may elect to ignore.

**$!INTERFACE INTERRUPTCHECKINGFREQUENCY =** *n*

Set the number of milliseconds between interrupt checks while Tecplot is drawing or processing data.

**$!INTERFACE THREEDVIEWCHANGEDRAWLEVEL = (FULL, TRACE)**

Set what to draw while Tecplot is performing 3-D view changes such as translating or rotating. You can choose **FULL** or **TRACE**.

**$!INTERFACE NONCURRENTFRAMEREDRAWLEVEL = (FULL, TRACE)**

Set how to draw all frames except the current frame. You can choose **FULL** or **TRACE**.

**31.1.3.2. OpenGL Specific Configuration Options.** The following options are available to further tune Tecplot to operate with the OpenGL capabilities of your platform. To assign values to these parameters you must use the **$!INTERFACE OPENGLCONFIG**.

**$!INTERFACE OPENGLCONFIG**

**{ SCREENRENDERING { DOEXTRADRAWFORLASTPIXEL = (YES, NO)}}**

Some OpenGL implementations use an optimization for line drawing that omits the last pixel in the line. Set this to **YES** to change all line drawing to force the last pixel to be drawn. This setting applies only to drawing on the screen.

**$!INTERFACE OPENGLCONFIG**

**{ SCREENRENDERING { STIPPLEALLLINES = (ALL, CRITICAL, NONE)}}**

Set to **ALL** to make all lines drawn using stippling. Set to **CRITICAL** to use stippling for stroke and user-defined fonts. Set to **NONE** to disable stippling. This setting applies only to drawing on the screen.

**$!INTERFACE OPENGLCONFIG**

**{ IMAGERENDERING { DOEXTRADRAWFORLASTPIXEL = (YES, NO)}}**

Some OpenGL implementations use an optimization for line drawing that omits the last pixel in the line. Set this to **YES** to change all line drawing to force the last pixel to be drawn. This setting applies only to exporting images from Tecplot.

**$!INTERFACE OPENGLCONFIG**

**{ IMAGERENDERING { STIPPLEALLLINES = (ALL, CRITICAL, NONE)}}**

Set to **ALL** to make all lines drawn using stippling. Set to **CRITICAL** to use stippling for stroke and user-defined fonts. Set to **NONE** to disable stippling. This setting applies exporting images from Tecplot.

### 31.1.4. Specifying Default File Name Extensions

The default extensions for file names in file input-output dialogs can also be changed in the configuration file. These settings are changed via the **FNAMEFILTER** sub-command in the **$!FILECONFIG** macro command.

- **COLORMAPFILE:** Specifies the default extension for color map files.
- **INPUTDATAFILE:** Specifies the default extension for input data files.
- **OUTPUTASCIIDATAFILE:** Specifies the default extension for ASCII output files.
- **OUTPUTBINARYDATAFILE:** Specifies the default extension for binary output files.
- **INPUTLAYOUTFILE:** Specifies the default extension for input layout and layout package files.
- **OUTPUTLAYOUTFILE:** Specifies the default extension for output layout files.
- **OUTPUTLAYOUTPACKAGEFILE:** Specifies the default extension for output layout package files.
- **STYLEFILE:** Specifies the default extension for stylesheet files.
- **MACROFILE:** Specifies the default extension for macro files.
- **EQUATIONFILE:** Specifies the default extension for equation files.

For example, to change the default extension for input data files to be **.tbl** use:

```
$!FILECONFIG
FNAMEFILTER
{
  INPUTDATAFILE = "*.tbl"
}
```

### 31.1.5. Specifying the Default Temporary Directory

Tecplot writes out a number of temporary files. To tell Tecplot where to place these files, put the following macro command in the **`tecplot.cfg`** file:

```
$!FILECONFIG
 TEMPFILEPATH = "tempfilepath"
```

where *tempfilepath* is the new path. The default path is system dependent.

## 31.2. Customizing Tecplot Interactively

Using the Preferences sub-menu under the File menu, you can interactively control the colors used throughout Tecplot, the size options available in most Tecplot dialogs, and several miscellaneous parameters.

### 31.2.1. The Color Preferences Dialog

To change the RGB values of Tecplot's basic colors, use the Color Preferences dialog, shown in Figure 31-2.

To change a color, click on it in the palette and alter its RGB values with the sliders. As you move the sliders, the box in the upper right corner of the dialog shows the color as currently specified. You may alter multiple colors by selecting those colors and changing their RGB values. Choosing Reset Selected Color or Reset All Colors will restore the default RGB values. All color changes take effect when you click OK.

### 31.2.2. The Size Preferences Dialog

To set size options, use the Size Preferences dialog, shown in Figure 31-2.

These options determine the choices available in drop-down menus such as Line Thickness that occur throughout the interface.

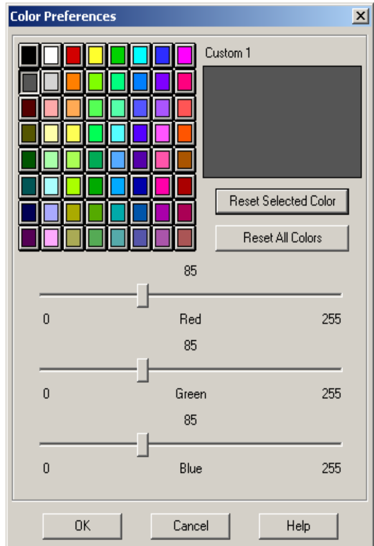



**Figure 31-2.** The Color Preferences dialog.

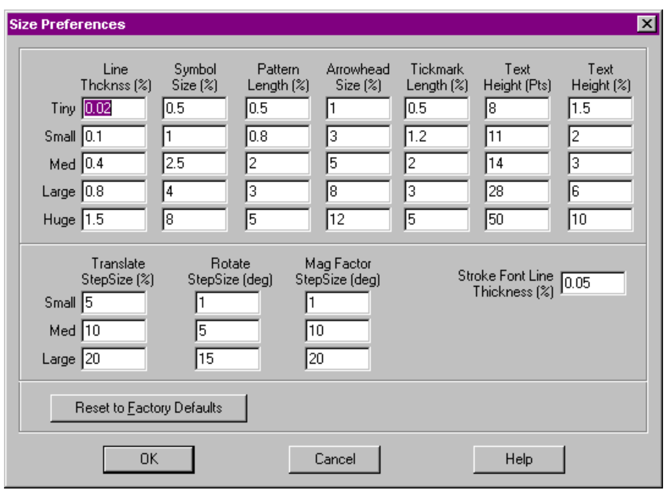



**Figure 31-3.** The Size Preferences dialog.

You can control the following sets of sizes:

- Line thickness.
- Symbol size.
- Pattern length.
- Arrowhead size.
- Tick mark length.
- Text height (in both points and frame units).
- Translate step size.
- Rotate step size.
- Magnification step size.
- Stroke font line thickness.

## 31.3. Using the Display Performance Dialog

Various aspects of what is displayed in the Tecplot workspace may be configured with the Display Performance dialog. Some of these options control the look of the workspace, while others enhance the graphics performance. The Display Performance dialog is shown in Figure 31-4.

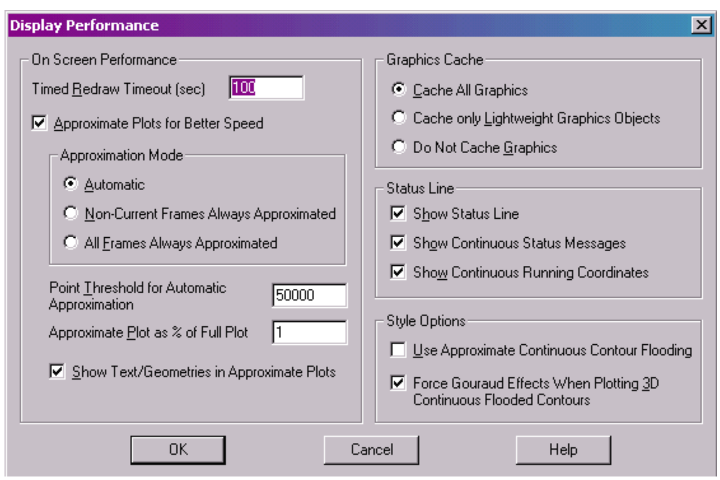



**Figure 31-4.** The Display Performance dialog.

There are many different combinations of computer systems and graphics card capabilities. Because of this no one setting is optimal for all hardware configurations. In addition, there are many different types of plots that can be made with Tecplot. Different plots can often be rendered faster if Tecplot is configured one way; others may be rendered faster if Tecplot is configured in a different way.

The Display Performance dialog allows you some control over how Tecplot manages the graphics resources. Often there is a trade off between level of detail or accuracy on the screen versus drawing speed.

For most general uses of Tecplot it is probably unnecessary to use the Display Performance dialog. However, if you are dealing with large data sets it is likely that you will want to make some adjustments.

The Display Performance dialog addresses four different areas:

- On screen performance.
- Graphics cache settings.
- Status line options.
- Other miscellanous style options.

The first two deal with changing Tecplot's performance by changing the level of detail of what is drawn or how often somthing is drawn. The second two merely allow you change what is seen in the work area or change the style for selected plot types to gain better performance.

### 31.3.1. On Screen Performance

Here you can configure the responsiveness of Tecplot and the level of detail.

**31.3.1.1. Timed Redraw Timeout.** Choose the number of seconds you want Tecplot to wait until it will automatically stops processing in preparation to draw. This should almost always be set to a high number. A drawing can always be interrupted by the user by clicking with the mouse in the work area

**Note:** There are a number of phases Tecplot goes through when processing a drawing in the work area. Most of the phases can be interrupted by clicking with the mouse in the work area. However, there are some phases that cannot be interrupted.

**31.3.1.2. Approximate Plots for Better Speed.** Selecting this option instructs Tecplot to render plots with a lesser degree of detail according to the Approximation Mode. The Approximation Mode can be one of the following:

**31.3.1.3. Automatic.** In this mode Tecplot will choose when the drawing in each frame is to be approximated based on the Point Threshold for Automatic Approximation. If the number of points used in a drawing in a frame exceed the Point Threshold then Tecplot will create an approximated version of the plot and render that first and then proceed to work on rendering a complete drawing. You will see the approximated plot quickly and you can at that point elect to make other changes to the plot. If you proceed in Tecplot before the final drawing is completed Tecplot will interrupt the processing of the detailed drawing. During rotation, translation, or smooth zoom, the plot will remain approximated until the mouse button is released.

**31.3.1.4. Non-Current Frames Always Approximated.** Choosing this option is like Automatic except that all non-current frames are always approximated. After a change in the current frame you will see all frames with an approximate drawing followed by the current frame drawn in full detail. The non-current frames are left as is. You can see a full detail drawing in all frames by clicking Redraw All on the sidebar or clicking with the middle mouse in the workarea (outside of any frame).

**31.3.1.5. All Frames Always Approximated.** In this mode the drawing in all frames is always approximated while you change views or style. You can see a full detail plot of the currently active frame by clicking Redraw on the sidebar or by clicking the middle mouse in the current frame. You can see a full detail plot of all frames by clicking Redraw All on the sidebar by clicking with the middle mouse in the work area (outside of any frame).

**31.3.1.6. Point Threshold for Automatic Approximation.** The drawing in a frame is a candidate to be approximated if the number of points used to draw the full plot exceeds this value. This is only a factor if the Approximation Mode is Automatic or Non-Current Frames Always Approximated.

**31.3.1.7. Approximate Plot as % of Full Plot.** When a plot is approximated it will be drawn with this percentage of the original data. In some cases Tecplot will plot more than this percentage to preserve the shape of the data.

**31.3.1.8. Show Text/Geometries in Approximate Plots.** Text and geometries are not approximated because it is assumed that they do not affect performance. If you are plotting a large number of text or geometries then you can unselect this option to remove text and geometries from the approximated views.

## 31.3.2. Graphics Cache

The Graphics Cache is used to store instructions for OpenGL display lists used to draw in the work area. If a significant amount of processing is needed to create the instructions, then saving the instructions for repeated use can increase the rendering speed significantly. The graphics cache can greatly speed up drawing for a number of situations including 3-D view

changes, changing the style in a sub-set of the plot (that is, changing the style of iso-surfaces only), or for repairing of the Tecplot work area after it has been damaged because a dialog has moved on top.

Some computers may have limited graphics resources compared to the amount of data to be plotted. In these cases it may be beneficial to either cache only "lightweight" graphics objects or do not cache anything at all.

If graphics are not cached you may experience situations where most or all of your plot is not drawn on the screen. This can occur when the work area is damaged by moving another dialog on top of it for example. If you have Auto Redraw turned off, the work area will have frames that are not drawn at all and remain that way until you click Redraw All.

**31.3.2.1. Status Line.** Three check boxes used to configure the status line are located at the top of the Performance Options dialog. You may enable or disable the display of running coordinates, status line messages, or turn off the status line altogether.

## 31.3.3. Style Options

This sections contains style options that can have an effect on the on-screen performance in Tecplot.

**31.3.3.1. Use Approximate Continuous Contour Flooding.** If this option is selected Tecplot will use a slightly faster but less accurate method for continuous contour flooding. With continuous contour flooding each cell is colored using a continuous variation across the cell from the colors derived at the nodes. In some cases the colors at opposing nodes in a cell may be different enough such that the colors derived for the middle of the cell may pass through the color cube and will extend outside of the colors defined in the Tecplot color map. This can produce things like a brownish color in the middle of a cell that has yellow and green on its edges. If this option is not selected, Tecplot takes extra care when it finds a cell where colors on opposing nodes straddle colors defined at the control points in the workspace color map. These cells are first subdivided to avoid this situation and then painted.

**31.3.3.2. Force Gouraud Effects When Plotting 3D Continuous Flooded Contours.** An unfortunate limitation in 3-D graphics causes the drawing of continous flooded contours on a surface that is panel shaded to be slow and very expensive in terms of the amount of resources needed to accomplish the task. In some cases the extra resources may overload the graphics capacity. Using Gouraud flooding in these cases is less taxing as well as faster. If this option is selected, Tecplot will automatically promote Panel shading to Gouraud shading when continuous contour flooding is used.

# 31.4. Configuring the Interface under UNIX

In UNIX, the style of the graphical user interface for Tecplot is configured for the most part by a resource file called **Tecplot90** which resides in the **app-defaults** sub-directory below the Tecplot home directory. If you edit this file the changes will affect all users. Alternatively, you can add entries to a file called **.Xdefaults** which resides in your own **$HOME** directory if you want the changes to apply only to your own execution of Tecplot. If the file **.Xdefaults** does not already exist in your home directory, you can create one.

## 31.4.1. Changing the Default Size of Tecplot

The resource lines that affect the default Tecplot process window size are:

```
*Tecplot.main_dialog.width: 900
*Tecplot.main_dialog.height: 720
```

Changing either the value 900 or the value 720 will change the default size of the Tecplot process window.

## 31.4.2. Changing Accelerator Keystrokes

Changing accelerator keystrokes is easiest using the following method:

1. Find the menu options to which you can assign accelerator keys in Tecplot. Use the **grep** command as follows:

   ```
   grep _mbopt.labelString Tecplot90
   ```

2. Find the accelerator keys not already assigned. The assigned keys can be found using the following command:

   ```
   grep accelerator Tecplot90
   ```

3. Add an entry to the **Tecplot90** or to the **.Xdefaults** file in your **$HOME** directory to add an accelerator. For example, the following line will let you press just the letter **N** to open a new layout:

   ```
   Tecplot*main_filenew_mbopt.accelerator: <Key>N
   ```

   The following example will let you press Ctrl-**N** to open a new layout:

   ```
   Tecplot*main_filenew_mbopt.accelerator: Ctrl<Key>N
   ```

### 31.4.3. Setting Default Positions for Dialogs

On UNIX platforms, you can customize the default positioning of modeless dialogs (those that can remain up while you do other things in Tecplot) by adding a few entries to your **`.Xdefaults`** file.

To change the default position for a given dialog, do the following:

1. Determine the base name of the dialog from table *xxxx* below.
2. Add the following lines to your **`.Xdefaults`** file:

   **Tecplot***_bbbbbb_**_dialog.defaultPosition: FALSE**

   **Tecplot***_bbbbbb_**_dialog.x:** *xxxxxx*

   **Tecplot***_bbbbbb_**_dialog.y:** *yyyyyy*

Where:

> *bbbbbb* is the base name of the dialog.
>
> *xxxxxx* is the X-position you want the dialog to start in.
>
> *yyyyyy* is the Y-position you want the dialog to start in.

The XY-position is relative to the upper left corner of the screen. Here are the base names of modeless dialogs in Tecplot.

| | | |
|---|---|---|
| **advancedbandopts** | **animaterecord** | **atprobe** |
| **axisedit** | **cellcenter** | **colormap** |
| **contcoloropts** | **contlabel** | **contlegend** |
| **contlinemode** | **contourlevels** | **contourvar** |
| **createcircularzone** | **createlinezone** | **createmirrorzone** |
| **createrectzone** | **createsubzone** | **dataalter** |
| **datalabels** | **datarotate2d** | **datasetinfo** |
| **dataspreadsheet** | **deletezone** | **depthblank** |
| **dupzone** | **editframe** | **editprobe** |
| **enterxyzone** | **extractcurrent3dslice** | **extractfebound** |
| **extractslice** | **frameaxis** | **geom** |
| **globalscat** | **globalstream** | **ijkblank** |
| **invdistinterp** | **isosurface** | **kriging** |
| **linearinterp** | **linkframes** | **macrorecord** |
| **macrovariables** | **macroview** | **optionstext** |
| **orderframes** | **paper** | **plotattr** |
| **polartorect** | **probe** | **quickedit** |
| **quickmacro** | **scatlegend** | **slicedetails** |
| **smooth** | **stream** | **text** |
| **threedadvanced** | **threedlightsrc** | **translate** |
| **triangulate** | **twod** | **valueblank** |

| | | |
|---|---|---|
| **vectorhead** | **vectorlength** | **view3d** |
| **viewrotate3d** | **xyeditprobe** | **xylegend** |

## 31.5. Defining Custom Characters and Symbols

When Tecplot starts up, it reads the font file (‘‘**tecplot.fnt**’’). This file contains information that defines the appearance of text characters on the screen. Tecplot defines and draws characters on the screen as a set of straight lines called strokes. These stroked characters approximate the appearance of characters for the screen.

The font file is an ASCII file that can be edited using an ASCII text editor. You can modify the shape, size, and resolution of existing stroke-font characters or add completely new ones. In PostScript print files, text characters are generated using PostScript defined fonts, not the stroked fonts. If you are using the Windows version of Tecplot and the Windows print drivers are active, then all text except text using the User-Defined fonts is serviced by the Windows printer driver. However, HP-GL and HP-GL/2 print files use the stroked fonts, and the text characters in bitmap export files are also in stroked fonts (since they are generated from the screen). The inter-character spacing in all output files is determined by the character-width definitions in the font file. When using PostScript print files or the Windows print drivers, changing the font commands affects only the character shape for User-Defined fonts and the character spacing for all fonts.

The Font File is structured as follows:

```
#!FF 4
 CharCellHeight
 Stroke command set for Helvetica Font
 Stroke command set for Greek Font
 Stroke command set for Math Font
 Stroke command set for User-Defined Font
 Stroke command set for Times Font
 Stroke command set for Times Italic Font
 Stroke command set for Courier Font
```

The file type and version are on the first line (‘‘FF’’ means Font File). *CharCellHeight* is the interline spacing (that is, the height of a capital M plus some vertical space) in the units of a two-dimensional coordinate system used to define the stroke-font characters. The baseline of the characters is at zero. Before Tecplot uses the character definitions, they are normalized by the character cell height.

Following the character cell height, there are seven sets of stroke commands, one set for each font as shown above. Each stroke command set consists of definitions for the characters in the font. Each font has a base set of 96 characters (character indices 32 to 127). Some fonts also

include an extended set of characters (character indices 160 to 255). The extended characters are needed to complete the character sets for most of the common European languages.

All seven stroke command sets must be present, and each must have at least one character defined. Each stroke command set begins with the definition for a space (character index 32). After that, characters within a stroke command set may be defined in any order. If a character is not defined in the Font File, it is drawn as a blank.

Each character in a stroke command set is defined as follows:

*CharIndex  NumCommands  CharWidth*
**`Command1`**
**`Command2`**
**`Command3`**
.
.
.
**`Command`***NumCommands*

*CharIndex* is the character index which ranges from 32 to 127 and 160 to 255 for each font (see Figure 18-5 for the matching of the character index to the English, Greek, Math, and standard User-Defined font characters), *NumCommands* is the number of stroke commands defining the character that follows, and *CharWidth* is the character width, which determines the spacing of the characters.

A command may be in one of the following forms:

- **m** *x y.*
- **d** *x y.*
- **mr** *dx dy.*
- **dr** *dx dy.*

A command that begins with an **`m`** is a move command. A command that begins with a **`d`** is a draw command. Commands **`mr`** and **`dr`** are relative move and relative draw commands. The *x* and *y* are the absolute coordinates within the character cell. The *dx* and *dy* are the relative coordinates with respect to the previous location (increments from the position attained by the previous command). All coordinates are specified as integers. Figure 31-5 shows an example of a character cell and the commands used to define the lowercase letter ‘‘y.’’ The height of the character cell is 48.

Figure 31-6 shows a symbol being defined. Symbols should be centered about (0, 0) so that they are centered about the point they mark. The font file included with Tecplot contains many User-Defined font stroke commands. Most of these are for creating extra plotting symbols,

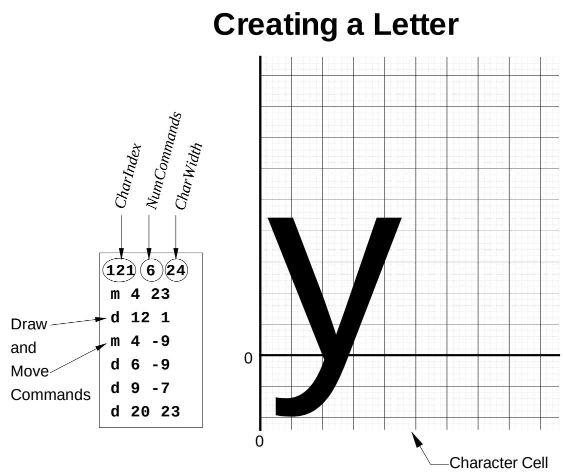



**Figure 31-5.** Defining a user-defined character.

accessible when you use the Symbol Type “Other,” enter an ASCII character, and specify the User-Defined font.

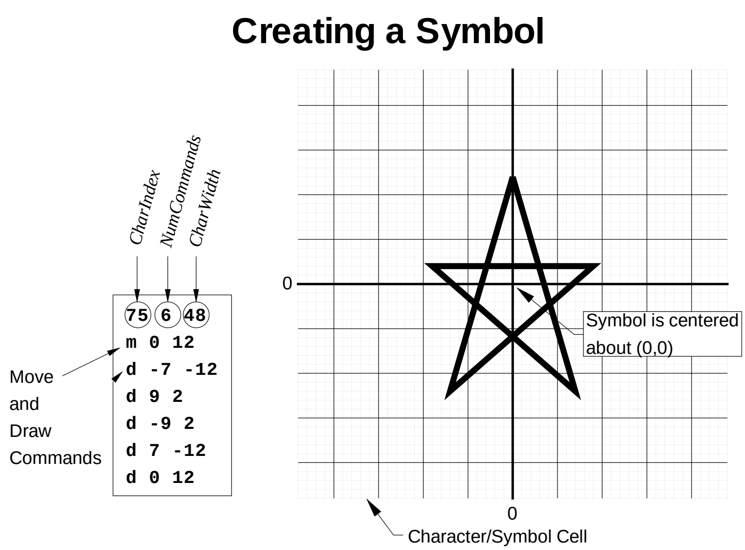



**Figure 31-6.** Defining a user-defined plotting symbol.

## 31.6. Configuring the Location of the "tecplot.phy" File

Whenever Tecplot starts, it tries to load a **`tecplot.phy`** file. This file contains information useful for running macros in batch mode (see Chapter 29, "Batch Processing," for more information) and also the name of the last layout file used in Tecplot. Whenever Tecplot exits, it writes out a new **`tecplot.phy`** file.

The place Tecplot looks for the **`tecplot.phy`** file is based on the following search:

1. Tecplot checks the environment variable **`TECPHYFILE`**. If this variable is set, Tecplot uses the value of this variable as the name of the **`tecplot.phy`** file. By default, this variable is not set. You can set this environment variable to control the location and name of the **`tecplot.phy`** file on a user-by-user basis.
2. (Windows Only) Tecplot checks the Windows registry for the key **`HKEY_LOCAL_MACHINE\SOFTWARE\Amtec Engineering, Inc.\Tecplot 9.0`**. If the value **`PhyFile`** is set under this key, then it is used as the name of the **`tecplot.phy`** file. This value is set by the installation program. You can use the command **`regedit`** from the Start Menu's Run option to edit the registry if you want to change or delete this key.
3. Tecplot uses the file called **`tecplot.phy`** in the directory where Tecplot is started. Note that this is the default behavior under UNIX.

Thus, using the default installation, Windows versions of Tecplot will write a **`tecplot.phy`** to one specific location (usually the Tecplot home directory), and UNIX versions will always use a **`tecplot.phy`** file in the directory where Tecplot is started.

The Windows version can be made to act like the UNIX version by deleting the value **`PhyFile`** from **`HKEY_LOCAL_MACHINE\SOFTWARE\Amtec Engineering, Inc.\Tecplot 9.0`** in the Windows registry with **`regedit`**.

Under both Windows and UNIX, the environment variable **`TECPHYFILE`** can be set to override this behavior.

# CHAPTER 32 Tecplot Add-Ons

Add-ons are a way to extend the basic functionality of Tecplot. They are executable modules designed to perform specific tasks. Amtec has produced a number of add-ons that load data in a variety of formats, allow advanced editing, or extend Tecplot's capabilities. By using the Tecplot *Add-on Developer's Kit* (ADK), users can create their own add-ons to generate plots, transform or analyze data, or perform a broad range of specialized tasks.

## 32.1. Using Add-Ons

Add-ons are external programs that attach themselves to Tecplot and are accessed through the Tecplot interface. When Tecplot is started, it goes through various initialization phases, including the processing of the **`tecplot.cfg`** file, the loading of the Tecplot stroke font file (**`tecplot.fnt`**) and the initialization of the graphics. After all of this has been completed, Tecplot begins to look for add-ons.

A number of the add-ons currently used by Tecplot are data file loaders or converters, which allow users to read non-Tecplot data files. These are:

- **loadplot3d:** A PLOT3D data loader.
- **loadxls:** An Excel file loader.
- **loadss:** A speadsheet file data loader.
- **loadgridgen:** A GridGen file data loader.
- **loaddxf:** A Data eXchange Format (DFX) data loader.
- **loadhdf:** A Hierarchical Data Format (HDF) data loader.
- **loaddem:** A Digital Elevation Map (DEM) data loader.
- **loadimg:** An add-on that loads bitmaps as a group of geometries.
- **loadcgns:** A CFD General Notation System (CGNS) data loader.
- **loadfluent:** A Fluent Version 5 data loader for **`.cas`** and **`.dat`** files.

These show up under the File drop-down under the Import option. The primary difference between loaders and converters are that loaders bring up more complex dialogs than do converters, which only bring up dialogs based on Tecplot's standard Load Data File(s) option.

Tecplot also uses add-ons for extended curve-fits with XY-plots. They may be accessed by selecting the Curve Type's Extended option, located on the Curve Attributes dialog.

Curve-fit add-ons include:

- **crvstineinterp:** A curve-fit using Stineman interpolation.
- **crvgen:** A curve fit where users define the equation.

Other add-ons may be accessed through the Tools drop-down on Tecplot's menu bar. They include:

- **advqet:** This calls up the Advanced Quick Edit dialog.
- **crsfez:** Allows extractions from finite-element sub-zones.
- **cstream:** Circle stream (this allows users to place a rake of streamtraces in a circular pattern).
- **statechange:** View all Tecplot state change information (used primarily for add-on development).
- **viewbin:** Binary data file viewer.

To use these add-ons, you may edit the **`tecplot.add`** file (located in the **`TEC90HOME`** directory), uncommenting the appropriate lines. For example, to use the advqet add-on, find the line which reads **`#$!LoadAddOn "advqet"`** in the **`tecplot.add`** file, remove the **`#`** sign, then save your changes. When you start Tecplot again, Advanced Quick Edit Tool will be an option under the Tools menu.

Finally, there are a number of add-ons related to Amtec's ADK (*Add-on Developer's Kit*). This consists of one add-on, **`GuiBuild`**, or Tecplot GUI Builder, along with several samples. These samples are add-ons which do not load automatically when you first install Tecplot.

If you want to build your own add-ons, you should refer to ADK documentation in the **`<tecplot-home-dir>/adk/doc`**. If this is not present, you may install the ADK by running the Tecplot installation program again, selecting to include the ADK during the installation.

### 32.1.1. Loading Add-Ons

You can customize lists of add-ons to be loaded by different Tecplot users in your network, or by a single user starting Tecplot with different commands.

**32.1.1.1. Add-Ons Loaded by All Users.** In a normal installation of Tecplot, the add-ons you want loaded by all users of Tecplot are named in an add-on load file called **`tecplot.add`**, located in the Tecplot home directory. The only command allowed in a **`tecplot.add`** file is the **`$!LoadAddOn`** command. The following is an example of a typical **`tecplot.add`** file:

```
#!MC 900
$!LoadAddOn "cfdtool"
$!LoadAddOn "streamtool"
```

**32.1.1.2. Specifying a Secondary Add-On Load File.** You may also instruct Tecplot to load a different list of add-ons by naming a second add-on load file using one of the following methods:

- Include **`-addonfile`** *addonfilename* on the command line.

or

- Set the environment variable **`TECADDONFILE`**.

Both of these methods tell Tecplot the name of another add-on load file to process.

**32.1.1.3. Specifying Add-Ons on the Command Line.** You can also instruct Tecplot to load a particular add-on via the command line. The following flags are available:

**`-loadaddon`** *libname*

or

**`-loadaxaddon`** *activeXname*

where

*libname*

The full name (including path and extension) of a **`V7Standard`** add-on (the only choice in UNIX).

*activeXname*

The name of an ActiveX style add-on. (The supplier of the add-on will tell you what type it is.)

You may specify the **`-loadaddon`** or **`-loadaxaddon`** flag as many times as you want on the command line.

If your add-on is named with the proper suffix for your platform (**`.dll`** for Windows, **`.sl`** for HP-UNIX, and **`.so`** for all other UNIX platforms) you can simply name the add-on on the command line without using the **`-loadaddon`** flag.

After add-ons are loaded, Tecplot re-processes all command line arguments not processed earlier (for graphics and add-on initialization). This ordering allows for a data reader add-on (discussed later) to be used to load data specified on the command line.

### 32.1.2. Using the $!LoadAddOn Command

The **`tecplot.add`** file is a special macro file that is executed at startup time and contains one or more **`$!LoadAddOn`** commands to load add-ons into Tecplot. **`$!LoadAddOn`** is, in fact, the only macro command allowed in a **`tecplot.add`** file. The syntax for the **`$!LoadAddOn`** command is:

**`$!LoadAddOn "`***libname***`"`**
  **`AddOnStyle =`** *addonstyle*

where

*libname*

The name of the shared object library file (see below). This must be in quotes.

*addonstyle*

The add-on style. This can be either **`V7Standard`** or **`V7ActiveX`**. **`V7Standard`** is the default.

Special rules govern how *libname* name is specified. In all cases the filename extension is omitted. If you assign *libname* to just the base name of the shared object library, then Tecplot will do the following:

- **UNIX:** The shared library to load will come from the file specified by:
  - *Tecplot-Home-Directory***`/lib/lib`**+*basename*+*platform-specific-extension*

where *platform-specific-extension* is **`.sl`** for HP platforms and **`.so`** for all others.

- **Windows:** If the add-on is of type **`V7Standard`** and just the base name is supplied, the add-on *basename***`.dll`** will be searched for in the following directories (in this order):
  - The directory where the Tecplot executable resides.
  - The Windows system directories.

- The directories in your **PATH** environment variable.

If an absolute path name is used in *libname*, then in Windows, **.dll** is appended and in UNIX **.so** or **.sl** is appended.

On Windows using **V7ActiveX** style add-on libraries, Tecplot connects to the add-on via the *libname* entry in the registry.

# APPENDIX A Tecplot Command Line Options

## A.1. Tecplot Command Line

The general form of the Tecplot command line is:

**`tecplot`** *[options] [layoutfile] [datafiles] [macrofile]*

where *options* is one or more of the following:

| | |
|---|---|
| **`-addonfile`** *filename* | Load add-ons listed in *filename.* |
| **`-b`** | Run Tecplot in batch mode (**`-p`** option is also required). |
| **`-c`** *cfgfile* | Use *cfgfile* for the configuration set up instead of the default configuration file. |
| **`-d`** or **`-display`** *computername* | Displays Tecplot on computer *computername* (UNIX only). The computer, *computername*, must have X-server capability with the GLX extension. |
| **`-datasetreader`** *readername* | Instruct Tecplot to use the data set reader *readername* when loading data files specified on the command line. See Section A.7, “Specifying Data Set Readers on the Command Line,” for details. |
| **`-debug`** *dbugfile* | Send debug information to the file *dbugfile*. Information is displayed to aid in debugging a new Tecplot configuration file, macro file, or binary data file. You may specify the minus sign (“**`-`**”) for *dbugfile* to send the debug output to the “standard output.” |
| **`-demo`** | Run Tecplot in demo mode (only reads demo files). |
| **`-develop`** | Launch Tecplot in a mode used to develop add-ons (UNIX only). |

| | |
|---|---|
| **-f** *fontfile* | Use *fontfile* for the font file instead of the default font file **tecplot.fnt**. |
| **-h** *homedir* | Use *homedir* for the Tecplot home directory instead of the default home directory or the directory stored in the operating system environment variable **TEC90HOME**. (See the *Tecplot Installation Notes*.) |
| **-loadaddon "***addonname***"** | Load add-on *addonname*. |
| **-loadaxaddon "***axaddonname***"** | Load Active-X add-on *axaddonname* (Windows only). |
| **-m** *cmapfile* | Select initial color map file to load. |
| **-n** | List node information (UNIX only). |
| **-nostdaddons** | Do not load add-ons in **tecplot.add**. |
| **-p** *macfile* | Play the macro in the file *macfile*. Note that if your macro file has an **.mcr** extension you do not need to use **-p**. |
| **-q** | Use quick playback mode. Ignores delay and pause commands. |
| **-qm** *quickpanelfile* | Load macro functions for the Quick Macro Panel from *quickpanelfile* instead of the default file **tecplot.mcr**. |
| **-r** *prtfile* | Set the default file name for routing Print Files to *prtfile*. This name can be reassigned interactively while running Tecplot. |
| **-s** *stylfile* | Use *stylfile* as a stylesheet for the first Tecplot frame. |
| **-showpanel** | Show the Quick Macro Panel immediately when Tecplot starts up. |
| **-v** | Print version number of Tecplot. |
| **-x** | Run Tecplot full screen. |
| **-y** *exportfile* | Same as **-r** except for exported files. |

In the command line, data files is one or more data files. These files are assigned to the first data set. You can also give the name of a layout file (typically having a "**.lay**" extension). Tecplot processes the layout immediately upon starting up. If both a layout file and data files appear on the command line, Tecplot substitutes the data files from the command line for the data files referenced in the layout file. When you read in a layout package file ("**.lpk**") you will not get this behavior.

# A.2. Using the Command Line in Windows

Most of the Tecplot command line options are available in Windows. To use them, you should start Tecplot from the Run command. In Windows the Run command is launched from the Start button. Under Windows you may also use the command line from the DOS prompt (known in Windows NT and Windows 2000 as the command prompt).

# A.3. Using Command Line Options in Windows Shortcuts

All of the command line options that can be entered at the DOS or Command prompt by using the Run command can also be used in a Windows shortcut.

## A.3.1. Creating Shortcuts

If you frequently run Tecplot using the same command line flags, it may be useful to create a shortcut on your Windows desktop that launches Tecplot with the desired command line flags. Here's how this can be done:

1. Right click in any blank space on your Windows desktop. A drop-down menu will appear.
2. Select New.
3. Select Shortcut from the next drop-down menu that appears.
4. The "Create Shortcut" dialog will appear (Figure A-1).

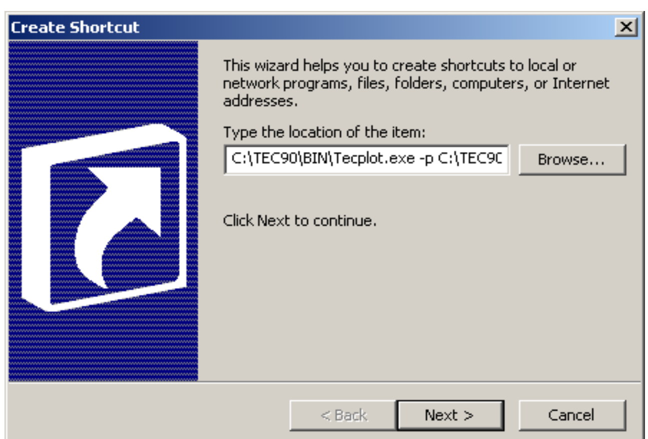



**Figure A-1.** The Create Shortcut dialog in Windows.

Type the location of the Tecplot executable, along with any command flags you want to specify. You can also click Browse if you are not sure where Tecplot is located. An example command line is:

```
C:\TEC90\BIN\Tecplot.exe -p C:\TEC90\mymacro.mcr
```

5. Click Next.
6. Select a name for your shortcut, then click on Finish. An example name would be:

```
Run my Macro
```

A new shortcut icon will be placed on your Windows desktop. To run Tecplot using the command line options you specified, simply double-click on the new shortcut icon.

## A.3.2. Changing Shortcuts

You can alter an existing shortcut by doing the following:

1. Right-click on the shortcut icon you want to change.
2. Select Properties from the drop-down menu.
3. On the Shortcut page (Figure A-2), modify the command line by changing the setting for Target. To change the working directory that Tecplot runs under, change the Start in location.

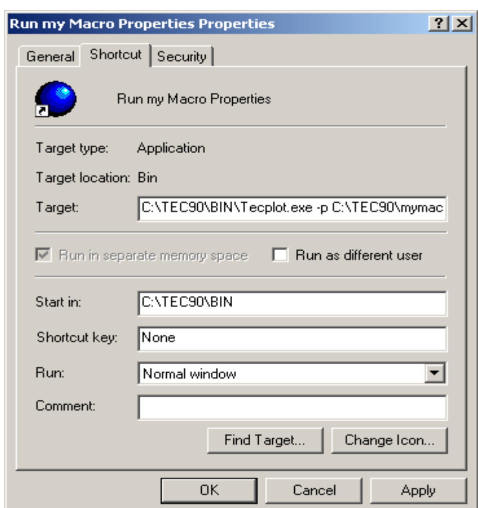



**Figure A-2.** The Shortcut page in Windows.

## A.4. Additional Command Line Options in Motif

Under UNIX, you can use additional command line flags which are passed to the window manager to control how the application window is displayed. These include **`-geometry`** (for specifying the location and position of the application window), **`-fg`** and **`-bg`** (for specifying foreground and background window colors), and others. See the X11 reference for your system for complete details on these options.

## A.5. Overriding the Data Sets in Layouts by Using "+" on the Command Line

This section describes how to load an alternate data set into a layout using the command line. A method for overriding the layout data set interactively, in the Open Layout dialog, is described in Section 5.1.2, “Loading Data from Other Software Packages.”

When loading a layout from the command line, you may override the data files used in that layout by specifying them on the command line after the layout file name. For example:

```
tecplot amt.lay t4.plt
```

This loads the **`amt.lay`** layout with the **`t4.plt`** data file instead of the data file specified in **`amt.lay`**.

If **`amt.lay`** had more than one data set associated with it, **`t4.plt`** would replace the first data set. If you wanted to replace multiple data sets, specify each file on the command line like so:

```
tecplot amt.lay t4-1.plt t4-2.plt t4-3.plt
```

This will use **`t4-1.plt`** as the first data set, **`t4-2.plt`** as the second, etc.

If **`amt.lay`**'s first data set had more than one data file associated with it, **`t4-1.plt`** would replace all data files in the first data set. If you wanted to specify more than one data file for the data set, you can use the **+** to specify they are all part of the first data set like so:

```
tecplot amt.lay t4-1.plt+t4-2.plt+t4-3.plt
```

In this case, **`t4-1.plt`**, **`t4-2.plt`**, and **`t4-3.plt`** are all combined into one data set that replaces the first data set of **`amt.lay`**.

You can combine both multiple data sets and multiple files per data set like so:

```
tecplot amt.lay ds1a.plt+ds1b.plt ds2.plt ds3a.plt+ds3b.plt
```

In this case, the files **ds1a.plt** and **ds1b.plt** are combined and replace the first data set, **ds2.plt** replaces the second data set, and **ds3a.plt** and **ds3b.plt** are combined to replace the third data set in **amt.lay**.

If you do not know which data set to substitute in your layout, look at the top of the layout file. It will look something like this:

```
#!MC 900
$!VarSet |LFDSFN1| = 'temp.plt'
$!VarSet |LFDSFN2| = 'chem.plt'
$!VarSet |LFDSFN3| = 'pos.plt'
```

So you can, for example, replace **chem.plt** with **chem1.plt** and **chem2.plt** using the following command line:

```
tecplot amt.lay temp.plt chem1.plt+chem2.plt pos.plt
```

## A.6. Tecplot Command Line Examples

To run Tecplot without pre-loading any data files, use:

```
tecplot
```

To run Tecplot loading the data file **ex1.plt** as the first data set, use:

```
tecplot ex1.plt
```

To run Tecplot loading the data files **ex1.plt**, **ex2.plt**, and **ex3.plt** as the first data set, use:

```
tecplot ex1.plt ex2.plt ex3.plt
```

To run Tecplot using **/usr/myhome** as the Tecplot home directory and loading the Tecplot configuration file **/usr/myhome/myset.cfg**, use:

```
tecplot -h /usr/myhome -c /usr/myhome/myset.cfg
```

To read a Tecplot layout file **sumtr1.lay**, you would use:

```
tecplot sumtr1.lay
```

To read a Tecplot layout file **calc.lay** and replace the first data set referenced in the layout file with the data file **temp.plt**, you would use:

```
tecplot calc.lay temp.plt
```

For example, suppose the layout file **t.lay** has two frames. The two frames reference different data sets. Suppose you want to start Tecplot, load this layout file, and have frame one use the data set defined in **a.plt** and have frame two use the data set defined by loading in **b.plt** and **c.plt** together. You can do this with the following command:

```
tecplot t.lay a.plt b.plt+c.plt
```

In UNIX, to determine the path or alias that the **tecplot** command calls, you would use:

```
which tecplot
```

# A.7. Specifying Data Set Readers on the Command Line

Special care should be taken when using the **-datasetreader** option on the command line. The following rules apply if **-datasetreader** is used:

1. The **-datasetreader** flag must be followed by the data set reader name and then immediately followed by a space separated list of commands to be passed on to the data set reader. No further Tecplot options are allowed after this point.
2. The data set reader name must be placed in quotes if it contains spaces.
3. Only one data set reader can be specified on the command line.
4. If a layout file is also specified (prior to **-datasetreader**) then you can only override the first data set load instructions referenced in the layout file.

Following is an example:

Suppose you have a layout file (**mylayout.lay**) that uses the PLOT3D loader. To launch Tecplot via the command line and override the PLOT3D load instructions use:

```
  tecplot mylayout.lay -datasetreader "plot3d loader" -ISET 1,,5 -b -3DW -GF
blunt.g
```

Everything from the **-ISET** parameter and following are instructions to be sent to the PLOT3D loader. Note that the instructions themselves are not entirely contained within any quotes. If your data reader requires instructions that themselves contain spaces then you must surround those instructions with quotes.

# APPENDIX B Utility Command Line Options

## B.1. Framer

To launch Framer at a command line (shell prompt, Run command, and so forth), use the following command:

**framer** *[options] [rmfile]*

where *[rmfile]* is the name of a file containing Raster Metafile bitmaps created by Tecplot, and *[options]* is one or more of the following:

| | |
|---|---|
| **-b [***nf***]** | Use buffered mode. Framer reads *nf* frames into memory and displays only those frames. Frames not read are not displayed. This mode displays images much faster, but requires extra memory. If *nf* is not specified, Framer reads as many frames as possible up to the total limit on frames (see **-max** parameter). |
| **-c** *nc* | Use no more than *nc* colors (X-Windows only). On some machines, you may need to use "**-c 128**" to allow two copies of Framer to run at the same time. |
| **-cycle** *nn* | Start Framer in "cycle" mode (as if **C** were pressed), and continue for *nn* complete cycles (unless interrupted by user input), and then exit. |
| **-d** *dfile* | Send debug information to *dfile*. Use "**-d2**," "**-d3**," "**-d4**,", etc., for more detailed debug information. |
| **-f** *start,end,skip* | Display frames starting with frame number *start* and ending with frame number *end*, skipping by *skip* frames. |
| **-g** | Use gray scale for image instead of color. |
| **-help** | Print help information. |

| | |
|---|---|
| **-loop** *nn* | Start Framer in "loop" mode (as if **L** were pressed), and continue for *nn* complete loops (unless interrupted by user input), and then exit. |
| **-m** | Allow for multiple color maps. Without this flag, Framer assumes the first color map in the Raster Metafile is valid for all images in that file. |
| **-max** *nn* | Specify upper limit on total number of images in the Raster Metafile. The default value is 512. |
| **-noinfo** | Do not print initial copyright notice, help info, or count of buffered frames. |
| **-p** *ms* | Pause at least ms milliseconds between each frame. This does not affect the rate of the single frame keys (**+** and **-**). |
| **-w** *wc* | Width correction. (Use "**-w -1**" for Tecplot Version 4 images.) |
| **-x** | Run full screen. |

If you do not specify a file name, Framer prompts you for one. You can choose to set buffering (equivalent to the **-b** flag) and/or multiple color maps (equivalent to the **-m** flag).

While Framer is running, you can press the following keys to control it:

| | |
|---|---|
| **B** | Move backward through frames (or left mouse button). |
| **C** | Cycle forward and backward through frames. |
| **F** | Move forward through frames (or middle mouse button on a three-button mouse or right mouse button on a two button mouse). |
| **L** | Loop repeatedly forward through frames. |
| **Q** | Quit Framer (or right mouse button on three button mouse) or Escape key. |
| **S** | Stop cycling or looping (or spacebar). |
| **R** | Redraw the current frame. |
| **1** | Move to the first frame. |
| **+** | Move forward one frame. |
| **-** | Move backward one frame. |
| **<** | Increase the minimum delay between frames by 50 milliseconds. This decreases the speed at which frames are displayed. |
| **>** | Decrease the minimum delay between frames by 50 milliseconds. This increases the speed at which frames are displayed. |

Under Windows, these Framer commands are also available from the Go and Step menus.

## B.2. LPKView

Following is a description of the utility's syntax. Brackets (**[]**) surround optional parameters and the vertical bar (**|**) separates one mutually exclusive set of options from another:

```
lpkview [[-t] | [-ild] | [[-c <preview command>] -p]] filename
```

where the options are described as follows:

| | |
|---|---|
| **-t** | Show table of contents. |
| **-i** | Extract image (for example, a Portable Network Graphics or **.png** format). |
| **-l** | Extract layout. |
| **-d** | Extract data. |
| **-c** | (UNIX only) Specify preview command. |
| **-p** | (UNIX only) Preview image. |

Option **-t** may not be used with any other options and options **-i**, **-l**, and **-d** may not be used with options **-c** and **-p**. If no command line options are specified **-i**, **-l**, and **-d** are assumed by default.

**Note:** Under UNIX, if the **-p** option is specified without specifying a preview command, **-c**, the following default preview command is used:

```
$MOZILLA_HOME/netscape -remote "OpenURL(%s)"
```

where **%s** is substituted by **lpkview** with the file name of the temporarily extracted preview image. The default command assumes that the environment variable **$MOZILLA_HOME** is set, Netscape is installed under **$MOZILLA_HOME**, and that **lpkview** has been added to Netscape as a helper application.

To add **lpkview** as a helper application bring up Netscape's Preferences dialog. This is usually accomplished by selecting Preferences from Netscape's Edit menu. Within the Preferences dialog locate and select the Applications page. Within the Applications page select New and add **lpkview** as a new helper application by entering the following information:

**Description:** *<optionally leave this blank>*

**MIMEType:** **application/x-tecplot-lpk**

**Suffixes:** **lpk**

**Handled By:** *<select "Application">*

**Application:** **$TEC90HOME/bin/lpkview -p %s**

Assuming that you have correctly set the **$TEC90HOME** environment variable, if you browse with Netscape and click on a layout package file, it will run **lpkview** as a helper application and display the preview image in your browser.

If you choose to specify your own preview command, there are several requirements:

- The path to the preview command must be fully specified.
- If relative, it must be located in one of the directories specified in your **$PATH** environment variable
- The command must contain a **%s** that can be substituted by **lpkview** with the file name of the temporarily extracted preview image.

For example, if you wanted the preview command to be the UNIX **file** utility. Then, running the following command:

```
lpkview -c "file %s" -p myplot.lpk
```

might produce the following output:

```
/var/tmp/aaaa005L7:    data
```

Where file **/var/tmp/aaaa005L7** is the temporarily extracted preview image. The temporary file is removed as soon as the preview command completes.

## B.3. Preplot

The following options are used with standard Tecplot data files:

| | |
|---|---|
| **-d** | Turn on debug echo. Use **-d2**, **-d3**, **-d4** for more detailed debug information. |
| **-r** | Reverse the bytes of the output binary data file (generally not required). |
| **-iset** *[zone]*, *[start]*, *[end]*, *[skip]* | |
| | Create the binary data file using only the specified range and skipping for the I-index. The arguments are optional, but the commas are not. The *zone* parameter specifies which zone this option affects; if not specified, all zones are affected. The *start* parameter is the starting I-index; the default is one. The *end* parameter is the ending I-index; the default is the last index value. The *skip* parameter specifies the I-interval, that is, the distance between indices; one means every index is used, two means every other index, and so on.<br><br>For example, **-iset 1, 3, 7, 2** indicates that for zone 1 only I-index values of **3**, **5**, and **7** are used. Only one **-iset** option is allowed per zone. |
| **-jset** *[zone]*, *[start]*, *[end]*, *[skip]* | |
| | Same as **-iset** above, except with respect to the J-index. |

| **-kset** *[zone]*, *[start]*, *[end]*, *[skip]* | |
|---|---|
| | Same as **-iset** above, except with respect to the K-index. |
| **-zonelist** *start[*:*end[*:*skip]]*, ... | |
| | Specify the zones to process. You may supply more than one specification. By default Preplot processes all zones. |

The following options are used with PLOT3D data files:

| | |
|---|---|
| **-d** | Turn on debug echo. Use **-d2**, **-d3**, **-d4** for more detailed debug information. |
| **-r** | Reverse the bytes of the output binary data file (generally not required). |
| **-plot3d** | Input file is in PLOT3D format. This flag is required for PLOT3D data. |
| **-b** | Input file is binary. |
| **-f** | Input file is binary-FORTRAN, that is, there are record markers. |
| **-foreign** | Reverse bytes of input file. |
| **-function** | The **.q** file is a **.f** file. |
| **-functionandq** | There are both **.f** and **.q** files present. |
| **-gridonly** | Read grid variables only. |
| **-i** | Input file includes PLOT3D IBLANK variable. |
| **-m** | Input file is multi-grid (usually more than one grid block). |
| **-ip** *ilist* | Extract planes of constant *i* for all *i* in *ilist*. (Requires 3-D whole data.) |
| **-jp** *jlist* | Extract planes of constant *j* for all *j* in *jlist*. (Requires 3-D whole data.) |
| **-kp** *klist* | Extract planes of constant *k* for all *k* in *klist*. (Requires 3-D whole data.) |
| **-1d** | Input PLOT3D file is 1-D. |
| **-2d** | Input PLOT3D file is 2-D. |
| **-3dp** | Input PLOT3D file is 3-D planar. |
| **-3dw** | Input PLOT3D file is 3-D whole. |

# B.4. Raster Metafile to AVI (rmtoavi)

The **rmtoavi** utility will convert a Raster Metafile animation to an AVI animation. The following is a description of the utility's syntax. Brackets (**[]**) surround optional parameters. Options must be specified separately.

> **rmtoavi** *[options]* **filename[.rm]** *[outputfilename]*

Filename is the name of the Raster Metafile to convert. Only one file name may be specified. The input file must end with the **.rm** extension.

The *[outputfilename]* is the name of the converted output AVI file. If the output file name is not specified, the input file name is used with an **.avi** extension. If any of the file names contain spaces, they must be enclosed in quotes.

For example, the command **rmtoavi test.rm** will create the file **test.avi**. If the output file exists, **rmtoavi** will prompt to overwrite it unless the **-y** option is used (see below).

The *[options]* are described as follows

| | |
|---|---|
| **-help** | Prints help information. |
| **-q** | Suppress startup banner and information message. |
| **-y** | Suppress query to overwrite an existing AVI file. |
| **-d [nn]** | Progress indicator. This prints a dot (**.**) every **[nn]** frames processed. If **[nn]** is not specified, it defaults to ten. |
| **-m** | Use multiple color palettes in the converted AVI file. Each frame of an AVI or Raster Metafile animation is limited to 256 colors. AVI animations can use either one set of 256 colors for the entire animation or a separate set of 256 colors for each frame. If you use the **-m** option, then each frame of the output AVI file will use a separate set of 256 colors. Since color information is read from the input Raster Metafile, this option only affects the output AVI animation if the Raster Metafile was originally exported using multiple color palettes. |
| **-speed nn** | Sets the speed of the output AVI file to *nn* frames per second. The default is ten. |

# APPENDIX C Mouse and Keyboard Operations

## C.1. Extended Mouse Operations

The middle and right mouse buttons are powerful tools you may use to immediately zoom and translate your data without having to switch to the Zoom or Translate tools on the sidebar. This advanced mouse/keyboard functionality is available when using any 3D rotate, Contour, Geometry (except Polyline), Probe, Slice, Streamtrace Placement, Translate, Zoom, or Zone Creation tools. If you have a two button mouse use the Ctrl key in conjunction with the right mouse button to achieve middle mouse button capabilities.

The following table lists all of the capabilities of the middle and right mouse buttons.

| Action | Middle Button/Ctrl-Right Button | Right Button |
|---|---|---|
| **`Click`** | Redraw. If the pointer is in the current frame then the current frame is redrawn. Otherwise, redraw all frames.[a] | Switch from the current tool to the Selector.[b] |
| **`Drag`** | Smoothly zoom in or out. An upward motion zooms out. A downward motion zooms in. | Translate. |
| **`Alt-Drag`** | In 3D frame mode, move the viewer further from (upward motion) or closer to (downward motion) the object. In all other frame modes, this behaves like the Drag action | Same as the Drag action. |

a. This is the default action for a click. It may be configured with the **`$!INTERFACE MOUSEMIDDLEBUTTONMODE`** command.

b. This is the default action for a click. It may be configured with the **`$!INTERFACE MOUSERIGHTBUTTONMODE`** command.

## C.2. Mouse Tool Operations

The following tables contain all mouse/keyboard operations you may use with the various side-bar tools. All mouse button operations utilize the left button.

3D Rotate tools:

| | |
|---|---|
| **`Drag`** | Rotate about the defined rotation origin with your current Rotate tool. |
| **`Alt-Drag`** | Rotate about the viewer position using your current Rotate tool. |
| **`C`** | Move rotation origin to probed point, ignoring zones. |
| **`O`** | Move rotation origin to probed point of data. |
| **`R`** | Rollerball rotation. |
| **`S`** | Spherical rotation. |
| **`T`** | Twist rotation. |
| **`X`** | X-axis rotation. |
| **`Y`** | Y-axis rotation. |
| **`Z`** | Z-axis rotation. |

Contour Add tool:

| | |
|---|---|
| **`Alt-Click`** | Place a contour line by probing on a streamtrace, slice, or iso-surface. |
| **`Click`** | Place a contour line. |
| **`Ctrl-Click`** | Replace the nearest contour line with a new line. |
| **`Drag`** | Move the new contour line. |
| **`-`** | Switch to the Contour Remove tool. |

Contour Remove tool:

| | |
|---|---|
| **`Click`** | Removes the contour line nearest to the probed location. |
| **`+`** | Switch to Contour Add tool if you are using Contour Remove. |

Geometry Polyline tool:

| | |
|---|---|
| **`A`** | Allow translation of polyline segments in all directions. |
| **`H`** | Restrict translation of current polyline segment to horizontal. |

| | |
|---|---|
| **U** | Pen up, while drawing polyline. |
| **V** | Restrict translation of current polyline segment to vertical. |

Probe tools.

| | |
|---|---|
| **Click** | If the pointer is over a valid cell return the interpolated field values from all nodes in the cell. If multiple cells are candidates then, for 2D frame mode the cell from the highest number zone is used and for 3D frame mode the cell closest to the viewer is used. |
| **Ctrl-Click** | If the pointer is over a valid cell return the field values from the nearest node in the cell. If multiple cells are candidates then, for 2D frame mode the cell from the highest number zone is used and for 3D frame mode the cell closest to the viewer is used. If the pointer is not over any cell then the field values from nearest data point as measured in distance on the screen are returned. |
| **Shift-Ctrl-Click** | Return the field values from the nearest point on the screen ignoring surfaces and regardless of zone number or depth of the point. This is useful in 3-D for probing on data points that are on the back side of a closed surface without having to rotate the object. In 2-D this is useful for probing on data points for zones that may be underneath other zones because of the order in which they were drawn. |
| **Alt-Click** | Same as Click except ignore zones while probing. (Probe only on streamtraces, iso-surfaces, or slices.) |
| **Alt-Ctrl-Click** | Same as Ctrl-Click except ignore zones while probing. (Probe only on streamtraces, iso-surfaces, or slices.) |
| **Alt-Ctrl-Shift-Click** | Same as Shift-Ctrl-Click except ignore zones while probing. (Probe only on streamtraces, iso-surfaces, or slices.) |

Slice tools:

| | |
|---|---|
| **+** | Turn on the start slice if no slices are active, or turn on the end slice if slices are already active. |
| **-** | Turn off the end slice if the end slice is active, or turn off the start slice if the end slice is not active. |
| **Click** | Place a start slice. |
| **Drag** | Move the start slice. |

| | |
|---|---|
| **Alt-click/Alt-drag** | Determine the XYZ-location by ignoring zones and looking only at derived volume objects (streamtraces, slices, iso-surfaces, slices). |
| **Shift-click** | Place the end slice. |
| **Shift-drag** | Move the end slice. |
| **I, J, K (ordered zones only)** | Switch to slicing constant I-, J-, or K-planes respectively. |
| **X, Y, Z** | Switch to slicing constant X-, Y-, or Z-planes respectively. |
| **0-9** | Numbers one through nine activate intermediate slices and set the number of intermediate slices to the number entered; zero turns off intermediate slices. |

Streamtrace Placement tools (3D Frame mode only):

| | |
|---|---|
| **D** | Switch to streamrods. |
| **R** | Switch to streamribbons. |
| **S** | Switch to surface lines. |
| **V** | Switch to volume lines. |
| **1-9** | Change the number of streamtraces to be added when placing a rake of streamtraces. |

Translate/Magnify tool:

| | |
|---|---|
| **Drag** | Translate the data. |
| **Shift-Drag** | Translate the paper. |
| **-** | If the drag was started with Shift, this will reduce the magnification of the paper. Otherwise, this will reduce the magnification of the data. |
| **+** | If the drag was started with Shift, this will increase the magnification of the paper. Otherwise, this will increase the magnification of the data. |
| **- drag** | Decrease magnification on the paper. |
| **+ drag** | Increase magnification on the paper. |

Zoom tool:

| | |
|---|---|
| **`Click`** | Center the zoom around the location of your click. |

## C.3. Picked Object Options

| | |
|---|---|
| **`-`** | Reduce the size of the object. If multiple objects are selected, all object positions will be shifted towards the first object selected. |
| **`-`** | Increase the size of the object. If multiple objects are selected, all object positions will be shifted away from the first object selected. |
| **`Del`** | Delete picked object(s). |
| **`Ctrl-C`** | Copy picked object(s) to the clipboard. |
| **`Ctrl-V`** | Paste picked object(s) from the clipboard. |
| **`Ctrl-X`** | Cut picked object(s). |

## C.4. Other Keyboard Operations

| | |
|---|---|
| **`Ctrl-A`** | Paste stored frame view to current frame. |
| **`Ctrl-C`** | Copy selected objects to paste buffer. |
| **`Ctrl-D`** | Redraw all frames. |
| **`Ctrl-F`** | Fit current image to full size. |
| **`Ctrl-L`** | Restore last frame view. |
| **`Ctrl-O`** | Open layout. |
| **`Ctrl-P`** | Print. |
| **`Ctrl-Q`** | Exit Tecplot. |
| **`Ctrl-R`** | Redraw the current frame. |
| **`Ctrl-S`** | Save current layout. |
| **`Ctrl-U`** | Call up the Publish dialog to control Web publishing. |
| **`Ctrl-W`** | Save current layout as a specified file. |

# APPENDIX D List of Example Files

These files can be found in the **TEC90HOME/demo/plt** and **TEC90HOME/examples/dat** directories. The file **all.lay** opens all **.plt** files in the **TEC90HOME/demo/plt** directory. The contents of any **.plt** file can be converted to a human-readable ASCII format by going to the File menu and selecting the Write Data File option. When writing out to an ASCII file, point format is preferable.

| File | .dat | .plt | XY | I (Irregular) | IJ (Surface) | IJK (Volume) | FE (Surface) | FE (Volume) |
|---|---|---|---|---|---|---|---|---|
| **3dfe** | | X | | | | | X | |
| **chem** | | X | X | X | | | | |
| **circle** | | X | | | X | | | |
| **create** | | X | X | X | X | | | |
| **creatvol** | | X | | | | X | | |
| **cstream** | | X | | | X | | | |
| **cylinder** | | X | | | X | | | |
| **dataltr** | | X | X | X | X | | | |
| **exchng** | | X | | | X | | | |
| **2dfed** | X | | | | | | X | |
| **3dgeom** | X | | | | X | | | |
| **febrfeb** | X | | | | | | | X |
| **febrfep** | X | | | | | | | X |
| **febrick** | | X | | | | | X | X |
| **feexchng** | | X | X | X | | | X | |
| **fetetbk** | X | | | | | | | X |
| **fetetpt** | X | | | | | | | X |
| **fetetra** | | X | | | | | | X |

| | | | | | | | | |
|---|---|---|---|---|---|---|---|---|
| **fetetra2** | | X | | | | | X | X |
| **fetriang** | | X | | | | | X | |
| **head** | | X | | | | | X | |
| **ijkcyl** | | X | | | | X | X | |
| **ijkortho** | | X | | | | X | | |
| **jetflow** | | X | | | X | X | | |
| **month** | | X | X | X | | | | |
| **month2** | | X | X | X | | | | |
| **movie** | X | | | | X | | | |
| **multzn** | | X | | | X | | | |
| **multzn2d** | X | | | | X | | | |
| **nozzle** | | X | | | X | | | |
| **polar2d** | | X | | | X | | | |
| **polar3d** | | X | | | | X | | |
| **position** | | X | X | X | | | | |
| **rain** | | X | X | X | | | | |
| **random** | | X | X | X | X | | | |
| **simp3dbk** | X | | | | | X | | |
| **simp3dpt** | X | | | | | X | | |
| **simpscat** | X | | | | X | | | |
| **simpxy** | X | | X | X | | | | |
| **simpxy2** | X | | X | X | | | | |
| **skirt** | | X | | | X | | | |
| **slice** | | X | | | | X | X | |
| **smooth** | | X | X | X | X | | | |
| **spcship** | | X | | | X | | | |
| **triang** | | X | X | X | | | X | |
| **twodrot** | | X | | | X | | | |
| **velocity** | | X | | | X | | | |
| **xtemp** | | X | X | X | | | | |

# APPENDIX E Glossary

The following terms are used throughout the *Tecplot User's Manual* and are included here for your information.

**2-D**

Plotting in two dimensions. Line plots of one or more variables (XY-plots) are not considered 2-D.

**2-D Field Plot**

A plot of some variable by location on a single plane using two axis. These plots are created in 2D frame mode.

**3-D**

Plotting in three dimensions. Three-dimensional plotting can be subdivided into 3-D surface and 3-D volume.

**3-D Field Plot**

A plot displaying a 3-D scattering of points, surfaces, or volumes using three axis. These plots are created in 3D frame mode.

**3-D Sorting**

The process by which Tecplot determines which surface to plot first. The various cells are sorted relative to the viewer and then plotted from farthest away to closest.

**3-D Surface**

Three-dimensional plotting confined to a surface. For example, the surface of a wing.

**3-D Volume**

Three-dimensional plotting of data that includes interior data points of a volume, as well as those on the surface. For example, the vector field around a wing.

**Active Zone**

A zone that is activated in the Plot Attributes dialog.

**ASCII Data File**

A data file composed of human-readable statements and numbers using ASCII characters.

**Aspect Ratio**

The ratio of lengths of the sides of an object. In 3D frame mode, the ratio is that of the longest side to the shortest side.

**Banded Contour Flooding**

A field plot where the surface between contour lines is filled with a constant color.

**Binary Data File**

A data file composed of machine-readable data. This type of file is created by converting ASCII data files with Preplot, or by directly creating them from an application.

**Blanking**

A feature of Tecplot that excludes certain cells and points from a plot. There are three types of blanking: value-blanked, IJK-blanking, and depth-blanking.

**Block**

A data file format in which the data is listed by variable. All the point values of the first variable are listed first, then all the point values of the second variable, and so forth.

**Boundary**

A 2- or 3-D field plot option. Plotting the boundary of a zone plots the connection of all outer lines (IJ-ordered zones), finite-element surface zones, or planes (IJK-ordered zones).

**Boundary Cell Faces**

A set of un-blanked cell faces in a 3-D volume zone which have only one neighboring volume cell. In contrast, interior cell faces have two neighboring volume cells, one on either side, which share the face. For an IJK-ordered zone the boundary cell faces are on the exterior of the zone. That is, the first and last I-planes, the first and last J-plans, and the first and last K-planes. For a finite-element 3-D volume zone, boundary cell faces are on the exterior of the zone and the surface of any voids within the zone.

**Bounding Box of Data**

The smallest rectangular box, aligned with the coordinate axes, which completely encloses all data points.

**Brick**

An element type of finite-element volume data composed of eight node points arranged in a hexahedron-like format. This element type is used in 3-D volume plotting.

**Carpet Plot**

A 3-D surface plot formed by a 3-D plot where the variable is plotted in the third dimension and is singular-valued with respect to the independent variables.

**Case Insensitive**

Text that may be in upper- or lowercase letters.

**Cell**

Either an element of finite-element data, or the space contained by one increment of each index of IJ- or IJK-ordered data.

**Color Map**

A color spectrum used to plot contour flooding and multi-colored objects.

**Color Map File**

A file that contains a description of a color map.

**Connectivity List**

The second portion of a finite-element data file where the relationships between points are given to define elements. Cells of the appropriate element type are defined by listing the node point indices. The number of node points per cell is determined by the element type.

**Continuous Contour Flooding**

A field plot where a color is assigned to each point in a mesh, based upon the contour variable and the color map. Each face is filled with colors interpolated between the corner nodes. This results in a smooth variation of color over the surface.

**Contour**

A field plot type that plots iso-valued lines, or color flooding based on the values of a specified variable.

**Curve Type**

The function used to fit the data points in an XY-plot.

**Custom Labels**

Text strings contained within a data file or text geometry file which define labels for your axes or contour table. You may select Custom Labels anywhere you can choose a number format, the result is the text strings in place of numbers.

**Cutaway Plot**

A 3-D volume plot where a portion of a 3-D volume zone is cut-away by blanking to reveal the interior.

**Cutting Plane**

A planar surface used to slice 3-D volume or surface zones.

**Data File**

A file that contains data used for plotting in Tecplot.

**Data Format**

The type of zone data as specified by the format parameter in a Tecplot data file, such as: BLOCK, POINT, FEBLOCK, or FEPOINT.

**Data Loader**

A Tecplot add-on which allows you to read non-Tecplot data files.

**Data Point**

An XYZ-point at which field variables are defined.

**Data Set**

A set of one or more zones. A data set may be plotted in one or more frames, however, a single frame may only plot one data set. A data set may be created by loading one or more data files.

**Dependent**

An axis mode requiring the axes to maintain a fixed ratio to one another.

**Depth**

For image export, the number of bits stored per pixel. For depth-blanking, the component of distance from the viewer position in a screen normal coordinate system.

**Depth-Blanking**

A blanking option which excludes cells in a 3-D plot, based upon their depth into the image. Cells closer than a plane of a certain depth, as well as cells further than a plane of another depth, may be blanked.

**Derived Volume Objects**

Graphic objects which are visible in the plot and created from zone data, but are not zones. Examples include iso-surfaces, 3-D slices, and streamtraces.

**Display List**

A group of OpenGL commands that have been stored for subsequent execution. Using display lists can, depending upon the hardware involved, dramatically speed up graphics rendering. Using display lists also requires more memory.

**Draw Level**

A draw behavior setting for modifying the image quality and rendering speed during various operations, such as rotation. Options vary from Trace, a simplified wire-frame mesh which is rendered quickly, to Full.

**Element Type**

The form of individual elements in a finite-element zone. There are four types: Triangle and Quadrilateral (finite-element surface types), and Tetrahedron and Brick (finite-element

volume types). The element type of a zone determines the number of nodes per element and their orientation within an element.

**Exposed Cell Faces**

The set of those cell faces in 3-D volume zones that have only one un-blanked neighboring volume cell. By comparison, interior cell faces have two neighboring cells, one on either side, which share the face. The exposed cell faces include boundary cell faces and interior cell faces exposed by blanking. (One of the neighboring cells has been blanked.)

**Extended Curve-Fit**

A Tecplot add-on which extends Tecplot’s XY-plot curve-fitting capabilities.

**Extra 3D Sorting**

Perform extra work to resolve hidden surface problems encountered during 3-D sorting.

**FE**

An abbreviation for finite-element, a common means of arranging data for calculations. (Often referred to as “unstructured.”)

**FEBLOCK**

A data file format for finite-element zones in which the node data is listed by variable. All the node values of the first variables are listed first, then the node values of the second variable, and so forth. This section is followed by a connectivity list.

**Fence Plot**

A plot of planes of a 3-D data field.

**FEPOINT**

A data file format for finite-element zones in which the node data is listed by point-by-point. All the variable values of the first point are listed first, then the variable values of the second point, and so forth. This section is followed by a connectivity list.

**FE Surface**

A finite-element zone of the element type Triangle or Quadrilateral. These zones are used for 2- and 3-D surface plots.

**FE Volume**

A finite-element zone of the element type Tetrahedron or Brick. These zones are used for 3-D volume plots.

**Field Layers**

One way of displaying a 2- or 3-D frame’s data set. The plot is the sum of the active zone layers, which may include mesh, contour, vector, shade, scatter and boundary.

**Field Plot**

Generally used to display the spacial relationship of data. These plots are created in 2D or 3D frame mode using any of the 2- or 3-D plotting options. Mesh, Contour, Vector, Scatter and Shade are all considered field plots. XY and Sketch frame modes are not field plots.

**File Path**

An option which specifies the directory for Tecplot to search for a given type of file. For instance, a linked layout saved with absolute file path contains the complete directory structure to load the associated file.

**Finite-Element**

A type of data point ordering. Data is arranged by listing the data points (called nodes), and then listing their relationships (called elements). The element type of the zone determines the number of nodes which are contained in each element, as well as the exact relationship of nodes within an element. There are four different element types supported by Tecplot: triangle, quadrilateral, tetrahedron and brick.

**Font Identifier**

The initial character used to embed Greek, Math, or User-Defined characters in a text string.

**Frame**

Boxed areas within the workspace where sketches and plots are created.

**Frame Mode**

Determines the type of plot which is displayed in a frame. For example, 2D field plot, 3D field plot, XY-plot, or Sketch plot.

**Geometry**

An arrangement of objects or parts that suggests geometric figures.

**Grid Area**

One or more rectangular regions defined and bounded by the grid axes.

**Grid Axes**

An axis option which displays the coordinates of the grid along the various spatial dimensions.

**Gridline**

A set of lines drawn from one or more axes that extend from the tick marks on an axis across the grid area.

**Grid Point**

In 2-D, the intersection of gridlines.

**I-Ordered**

A type of data point ordering where each point is listed one at a time (that is, by one index). Used mainly in XY-plots. In 2- or 3-D, this type of data point ordering is sometimes called irregular, and is only useful for scatter plots, or for interpolating or triangulating into 2-D, 3-D surface, or 3-D volume zones. (This type of data can also be used for 2- or 3-D vector plots if streamtraces are not required.)

**IJ-Ordered**

A type of data point ordering where the points are arranged in a 2-D array. used for 2-D and 3-D surface plotting.

**IJK-Blanking**

A feature to include or exclude portions of an IJK-ordered zone based on index ranges.

**IJK-Ordered**

A type of data ordering where the points are arranged in a 3-D array. Used for 3-D volume plotting as well as 2-D and 3-D surface plotting.

**Image Format**

Any of the raster or bit-mapped graphic formats supported by Tecplot.

**Inactive Zone**

A zone loaded into Tecplot which does not appear in the plot. A zone can be deactivated using the Zone Show option on any page of the Plot Attributes dialog.

**Independent**

Axis mode allowing each axis to have a range that is not affected by the ranges of other axis or axes.

**Interpolate**

To assign new values for the variables at data points in one zone based on the data point values in another zone (or set of zones).

**Internal Macro Variable**

A read-only macro variable which allows you to access certain key values in Tecplot. For example, **`$NUMVARS`** gives the number of variables.

**I-Plane**

In an ordered zone, the connected surface of all points with a constant I-index. In reality, I-planes may be cylinders, spheres, or any other shape.

**Irregular Data**

Points which have no order, or at least no order which can be easily converted to IJ- or IJK-ordering.

**Iso-Surface**

A surface within a 3-D zone where the contour variable has a constant value at all locations.

**J-Plane**

In an ordered zone, the connected surface of all points with a constant J-index. In reality, J-planes may be cylinders, spheres, or any other shape.

**K-Plane**

In an IJK-ordered zone, the connected surface of all points with a constant K-index. In reality, K-planes may be cylinders, spheres, or any other shape.

**Layout File**

A specialized macro file which preserves a plot created within Tecplot. When the layout is opened, it restores Tecplot to the state it was in when the layout file was saved.

**Layout Package File**

A binary layout file with the data embedded.

**Macro**

A file containing a list of instructions, called macro commands, which can duplicate virtually any action performed in Tecplot.

**Macro Command**

An instruction given to Tecplot in a macro file. Macro commands always start with a dollar sign and then an exclamation mark. For example, $!Redraw refreshes a plot view.

**Macro File**

A file which contains a series of macro commands. Macro files are run from the command line, or through the Run option of the Macro sub-menu of the File menu.

**Macro Function**

A self-contained macro sub-routine that can be called.

**Macro Variable**

A holding place for numeric values in a macro file. There are two types of macro variables: user-defined (you set and retrieve the value), or internal (Tecplot sets the value and you may retrieve it).

**Median Axis**

In 3-D, the grid axis which when scaled is not the shortest nor the longest axis.

**Menu Bar**

The top bar of the Tecplot screen used to select menu options.

**Mesh**

A 2- or 3-D field plot type which plots connections between data points.

**Multi-Colored**

Any Tecplot object which is colored by the value of the contouring variable. Multi-colored objects may include mesh, scatter symbols, vectors, contour lines, and streamtraces.

**Multi-Line Text**

Text which spans two or more lines.

**Node**

A point in finite-element data.

**Number Format**

The style of numbers to display for a data or axis label; exponent, integer, float, and so forth.

**OpenGL**

A graphics library for high-end 3-D graphics. It usually takes advantage of hardware acceleration for 3-D rendering.

**Ordered Data**

A type of data point organization which consists of a parameterized series of points. There are seven types of ordered data: I-, J-, K-, IJ-, JK-, IK-, and IJK-ordered. I-, IJ-, and IJK-ordered are the most common.

**PLOT3D**

A plotting package developed by NASA. Useful because the file format can be converted to a Tecplot binary data file by Preplot.

**Point**

A data file format for an I-, IJ-, or IJK-ordered zone in which the data is listed by point. All of the variable values for the first data point are listed first, then all the variable values for the second data point, and so forth.

**Precise Dot Grid**

In 2-D, the points of intersection of the imaginary lines extending from the X- and Y-axes' tick marks.

**Preview Image**

A display of your plot as it will appear when printed.

**Primary Corner**

The point in an ordered zone's cell that has the minimum index values for that cell, or the first listed node of a finite-element cell.

**Print File**

An output file which contains a description of the plot. (Used for making hard copies.)

**Print Format**

The type of print output. For example. PostScript, HP-GL/2, and so forth.

**Quadrilateral**

An element type of finite-element surface data which is composed of four node points arranged in a quadrilateral. Used in 2- and 3-D surface plotting.

**Quick Macro Panel**

A user-defined panel accessed from the Tool menu which allows quick access to your macro functions.

**Rake**

A specified line from which two or more streamtraces are generated.

**Ribbon**

(See *Streamribbon*.)

**Rod**

(See *Streamrod*.)

**Scatter**

A 2- or 3-D field plot type which plots a symbol at each data point.

**Shade Plot**

A 2- or 3-D field plot type which plots solid color or colors with lighting effect over the cells of the data.

**Sidebar**

The area to the left of the Tecplot workspace.

**Sketch Plot**

A plot which displays only text and geometries. These plots are in Sketch frame mode.

**Slice**

A set of data created by the intersection of a plane with 3-D zones.

**Snap-to-Grid**

Lock any object on the screen to the closest grid point. The position and size of the object will be affected by changes to the grid.

**Snap-to-Paper**

Lock any object on the screen to the underlying paper. The position and size of the object will not be affected by changes to the grid.

**Sort**

A measurement from one to two of the amount of work Tecplot should do to resolve hidden-surface problems during 3-D sorting. Selecting two will increase the time required for each redraw and will generate messages about the number of cells with a potential conflict.

**Step Size**

The fraction of a cell over which Tecplot streamtraces are integrated. Step size in Tecplot is variable, changing with the vector field and the size and aspect ratio of the cells.

**Stream**

An option of vector plots to plot particle traces through the vector field.

**Stream Format**

The current type of streamtraces being placed in Tecplot. For example, Surface Line, Volume Line, Volume Ribbon, or Volume Rod.

**Streamline**

A 2- or 3-D line which is parallel to the vector filed along its entire length. For a steady state vector field, this is the same as a simple particle trace which marks the path of a massless particle in the vector field.

**Streamribbon**

A particle trace with a width which not only follows the flow field (its center being a regular streamline), but which also twists with the vorticity of the vector field.

**Streamrod**

A particle trace with a polygonal cross-section and a width which not only follows the flow field (its center being a regular streamline), but which also rotates with the vorticity of the vector field.

**Streamtrace**

Any type of particle trace: streamlines, streamribbons, or streamrods.

**Streamtrace Zone**

Any streamtrace which has been extracted to form a new zone.

**Stylesheet**

A type of file which contains the definition of how the plot in a single frame is to be plotted. The stylesheet does not contain any zone data but does contain information about views, axes positions, zone attributes, and so forth.

**Surface Line**

A type of 3-D streamline which is confined to remain on a 3-D surface. Also used to refer to 2-D streamlines.

**Tetrahedron**

An element type of finite-element surface data which is composed of four node points arranged in a tetrahedron. (Used in 3-D volume plotting.)

**Translucency**

A property allowing you to see through an object to areas within or beyond it. In Tecplot you may vary the amount of translucency, controlling the extent that an object closer to you obscures one it overlays.

**Triangle**

An element type of finite-element surface data which is composed of three node points arranged in a triangle. (Used in 2- and 3-D surface plotting.)

**Unordered Data**

(See *Irregular Data*.)

**Value-Blanking**

A feature of Tecplot used to trim or eliminate cells based on one or more user-defined constraints for variable values.

**Variable**

One of the values defined at every data point in a Tecplot data set or data file.

**Vector**

A short line or arrow showing the direction and or the magnitude of vector qualities.

**Volume Line**

A type of 3-D streamline which is not confined to remain on a surface and may travel through 3-D volume data.

**Volume Zone**

Any zone that is IJK-ordered, finite-element tetrahedron, or finite-element brick.

**Vorticity**

The measurement of the tendency of a vector field to rotate about a point. (Also called “curl.”)

**Workspace**

The portion of your screen where you can create Tecplot frames. This includes but is not limited to the region covered by the displayed paper.

**XY-Dependent**

A 3-D axis mode where X and Y are fixed (dependent), but Z is free to vary in ratio (independent).

**XY-Map**

A set of points from a single zone where one variable is assigned to an X-axis and another is assigned to a Y-axis. You can define many XY-maps for an XY-plot.

**XY Map Layer**

One way of displaying an XY-map, such as with line, bars, symbols, and so forth. One XY-map may be displayed with one or more layers.

**XY-Plot**

Plots one variable assigned to one axis versus another variable assigned to another axis. Log plots, bar charts, curve fitted lines are all examples of XY-plots. These plots are created with XY frame mode.

**Zone**

A subset of a data set which is assigned certain plot types. Zones may be activated (plotted) or deactivated (not plotted). Each zone has one type of data ordering: I-, IJ-, IJK-, or finite-element. Zones are typically used to distinguish different portions of the data. For example, different calculations, experimental versus theoretical results, different time steps, or different types of objects, such as a wing surface versus a vector field around a wing.

# APPENDIX F Limits of Tecplot Version 9.0

The following hard limits apply to Tecplot Version 9.0.

| Item | Limit |
|---|---|
| Maximum number of data points per variable | Over 2 billion |
| Maximum number of zones per data set | 32,700 |
| Maximum number of variables per data set | 32,700 |
| Maximum number of XY-mappings | 32,700 |
| Largest floating point absolute value | $10^{150}$ |
| Smallest non-zero floating point absolute value | $10^{-150}$ |
| Maximum number of picked objects | 1500 |
| Maximum number of data sets | 128 (Limited by max. number of frames) |
| Maximum number of frames | 128 |
| Maximum number of value blank constraints | 8 |
| Maximum number of geometries | limited by memory |
| Maximum number of polylines per line geometry[a] | 50 |
| Maximum number of points per circle or ellipse | 720 |
| Maximum number of custom label sets | 10 |
| Maximum number of custom labels per set | 5000 |
| Minimum frame width or height | 0.1 inches |
| Maximum frame width | 500 inches |

| Item | Limit |
|---|---|
| Maximum streamtraces per frame | 5000 |
| Maximum number of color map overrides | 16 |
| Maximum preview width for EPS files | 1024 |
| Maximum preview height for EPS files | 1024 |
| Maximum number of user-defined color map control points | 9 |
| Maximum number of raw user-defined color map entries | 800 |
| Maximum number of characters in variable name | 64 |
| Maximum number of characters in zone title | 64 |
| Maximum number of characters in data set title | 256 |
| Maximum number of views per view stack | 16 |

a. A polyline is a continuous series of line segments, and can be a subset of a line geometry.

The following soft limits may be changed via the Tecplot configuration file:

| Number of: | Windows | UNIX | Hard Limit |
|---|---|---|---|
| Points per line[a] | 3000 | 5000 | 500,000 |
| Contour levels | 150 | 400 | 5000 |
| Characters per text label | 1023 | 1023 | 10,000 |

a. Points per line is the limit on the number of points allowed in the following: line segment geometries, stream termination lines, and contour lines. For line segment geometries, this is the total number of points used in all polylines contained in the geometry.

The following hard limits apply to plot style:

| Item | Limit |
| --- | --- |
| Printing Gouraud shaded plots with continuous flooding | On screen or exported bitmap image only |
| Printing plots with translucency | On screen or exported bitmap image only |

# Index

## Symbols



## Numerics

### D

**E**

## G

## H

## I

### J



### K

**N**

**W**

**X**

## Y



## Z